UNIT 11: Revolved and Extruded Shapes

In addition to basic geometric shapes and importing of three-dimensional STL files, SOLIDCast allows you to create three-dimensional shapes that are formed by revolving or extruding two-dimensional shapes (flat cross sections).

These two-dimensional shapes may be contained in FILES, or alternatively you can SKETCH shapes on the screen, using the mouse.

If you are importing two-dimensional shapes from FILES, these files can be either of two types:

1. AFSCad files
2. DXF files

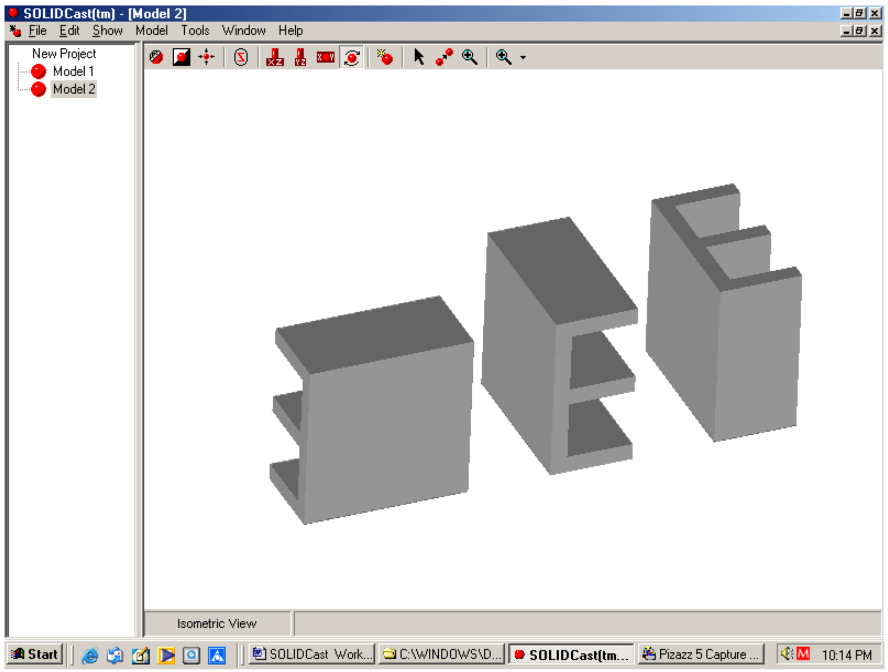
AFSCad files are drawing files that are created by the AFSCad program, which is a 2D drawing program included with Version 4.20 of the AFS Solidification System (3D), which was the DOS forerunner of SOLIDCast. While not a Windows application, AFSCad can be run in a DOS window using a product like DOSBox.

DXF files are drawing files that can be created by almost any CAD system. To use a DXF file, you would create the 2D cross section of the casting in your CAD system and save this as a DXF file, then import the file into SOLIDCast to create a solid of revolution or extrusion. SOLIDCast will interpret DXF files that have been created using lines, circles and arcs.

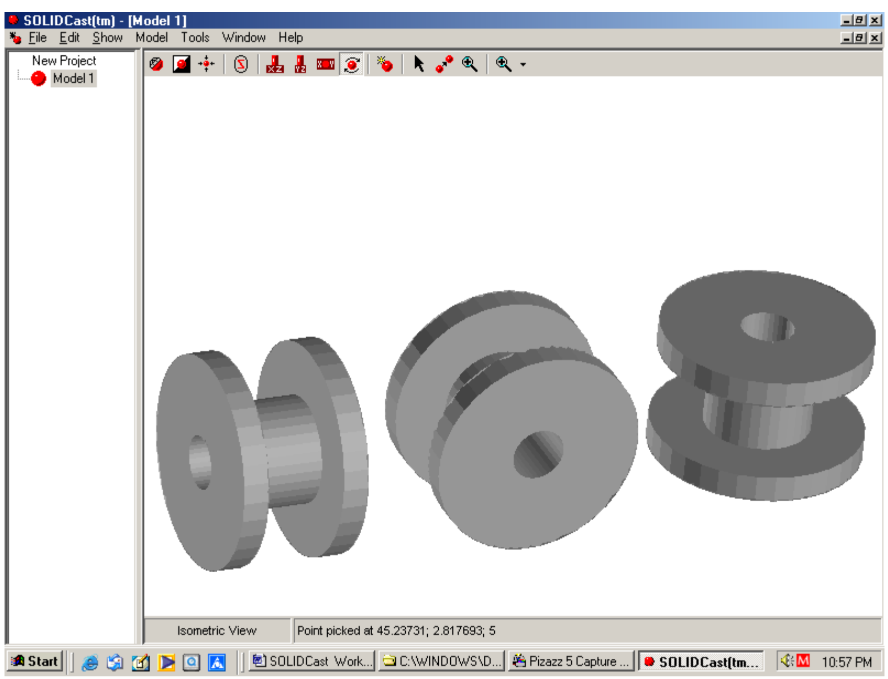
Note that when you import a 2D shape, the coordinates of that 2D shape determine where it will be located in 3D space. For example, if you have a circle centered on the point (X=2.5,Y=4.0) in a 2D drawing, and then import this drawing to make an extrusion in the Z direction, the resulting shape will still be centered on the point (X=2.5,Y=4.0) in 3D space. This is how objects can be correctly located when they are imported into SOLIDCast for extrusion or revolution.

A drawing file (whether in AFSCad or DXF format) must contain ONLY ONE complete cross section for the purpose of creating a solid of revolution or extrusion. The cross section must be fully enclosed, i.e., there should be no gaps or crossovers in the cross section. If you have multiple 2D shapes that are required in order to make a complete casting model, each 2D shape must be stored in a different file.

An extrusion refers to the creation of a shape that has a constant cross section. The ends of the extrusion are parallel. In SOLIDCast, you can create an extrusion parallel to the X axis, the Y axis or the Z axis. Examples of each are shown here:



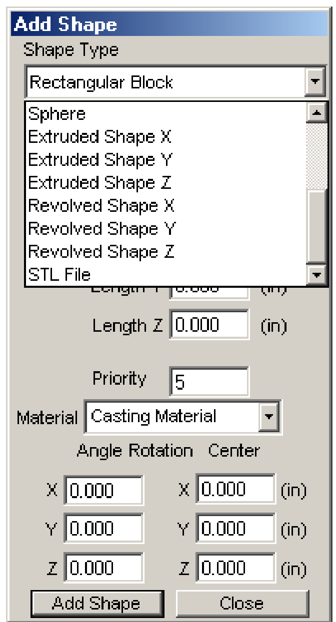
Solids of revolution are created by rotating a two-dimensional cross section about an axis of revolution. The axis may be parallel to the X axis, the Y axis or the Z axis, as shown here:



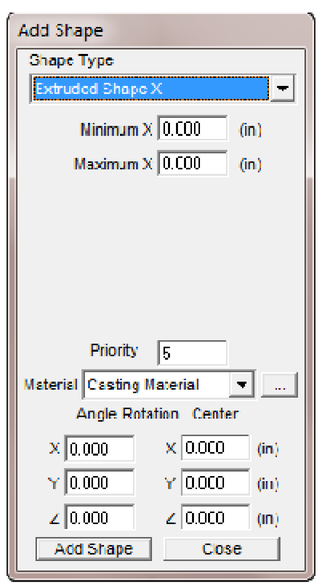
Creating Extruded Shapes from AFSCad and DXF Files

To create a solid of extrusion, you must have a model displayed or a blank model space showing (select Model…New Model from the main menu to create a blank model space). Then click on the icon that says "Add a shape to the model" (you can see the icon labels if you pass the mouse over the icons on the toolbar). You will see the Add Shape window appear.

Click on the "down arrow" and scroll down the list until the extruded shapes appear as follows:



If you select one of the extruded shapes, you will see the following appear:



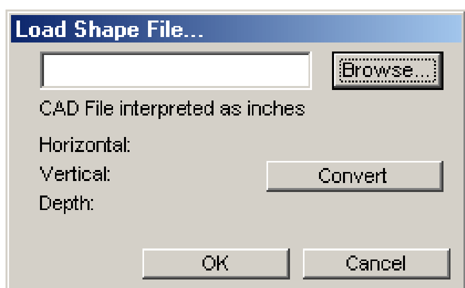
Extruded Shape X has been selected. This indicates that a 2D shape will be extruded parallel to the X axis. The system wants to know the length of the extrusion, which is given by entering the X value at the start of the extruded shape, and the X value at the end of the extruded shape. For example, if an extrusion starts at X=1.5 and ends at X=10, then the length of the extruded shape will be 8.5.

When an Extruded Shape X is selected, the system will automatically adjust the view so that you are viewing the model in the YZ view. When selecting Extruded Shape, the view is automatically adjusted as follows:

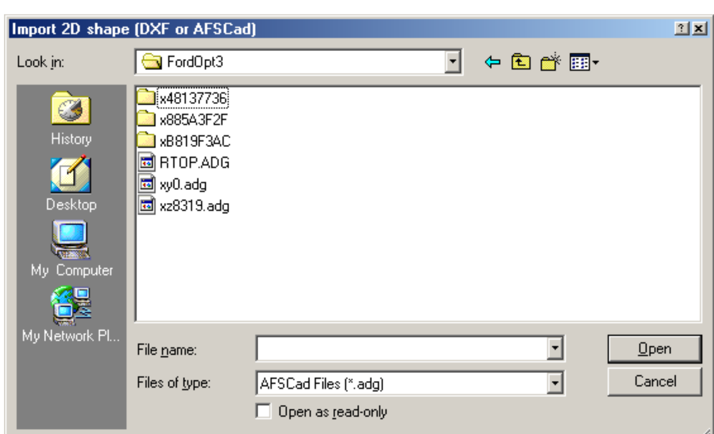
Extruded Shape X	YZ View
Extruded Shape Y	XZ View
Extruded Shape Z	XY View

You will notice that, when you select Extruded Shape X, the toolbar at the top of the screen changes and appears as follows:

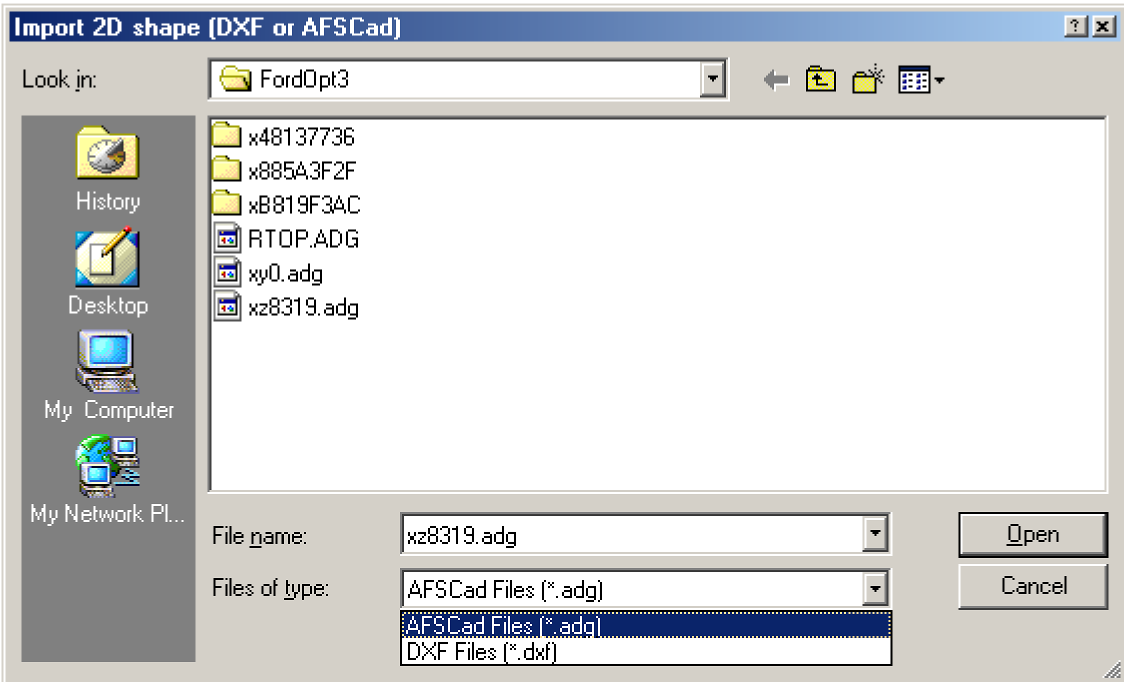
In order to indicate to the system that you will be loading a 2D file (either a DXF file or an AFSCad drawing file) click on the toolbar button labeled “dxf/adg”. You will see a window that appears as follows:



This window now allows you to select a file (DXF or AFSCad) from which to load the 2D shape. Click on the Browse button on this window. Another window will open which lets you select an AFSCad file (or a DXF file), as follows:

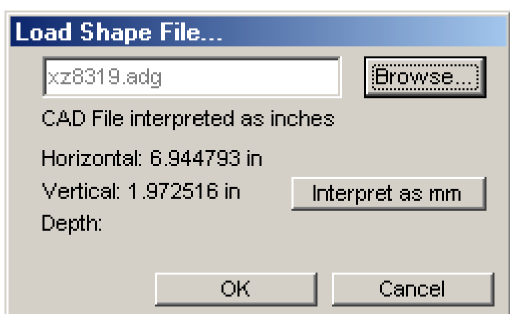


This window initially opens into the folder that is specified under System Parameters for Import Files. You can search through different folders using standard Windows navigating techniques. You can also search for DXF files by clicking on the "down arrow" next to the label "Files of type", as shown here:



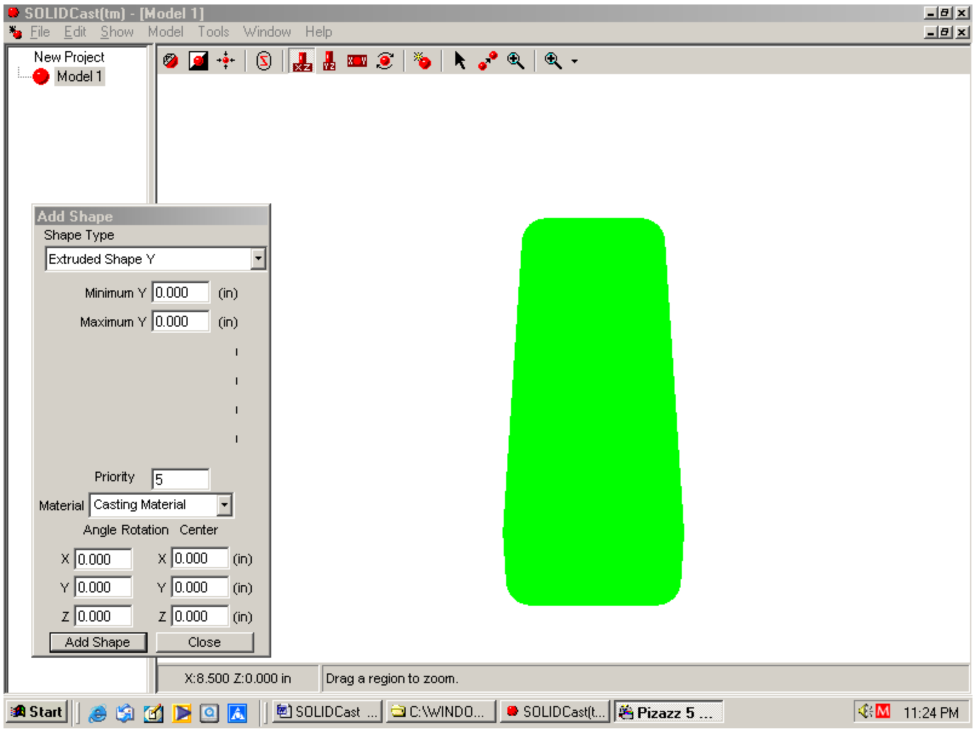
By selecting DXF Files you can then browse through any DXF files that you want to load.

After highlighting a file, click on the Open button. This will then bring you back to the preceding window, and it will appear similar to the following:

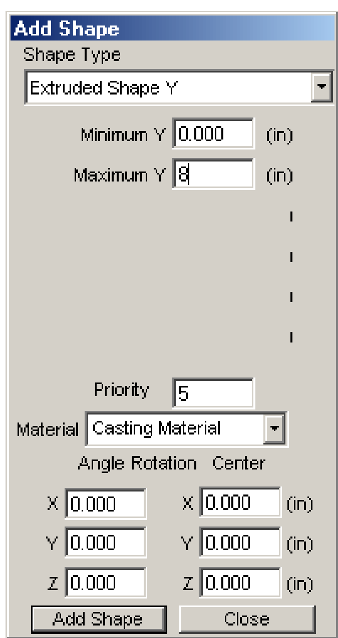


The overall dimensions of the shape (in two directions) are given on this screen. The system displays (in inches or millimeters) what it believes to be the dimensions of this shape. If the dimensions are incorrect, it may be because the shape was saved in millimeters and the system is interpreting the dimensions as inches, or vice versa. In this case, you can click on the button labeled "Interpret as mm" and the dimensions will be converted. When this occurs, the button label changes to "Interpret as inches". Each time you click this button, it toggles back and forth between interpreting the file as inches or mm.

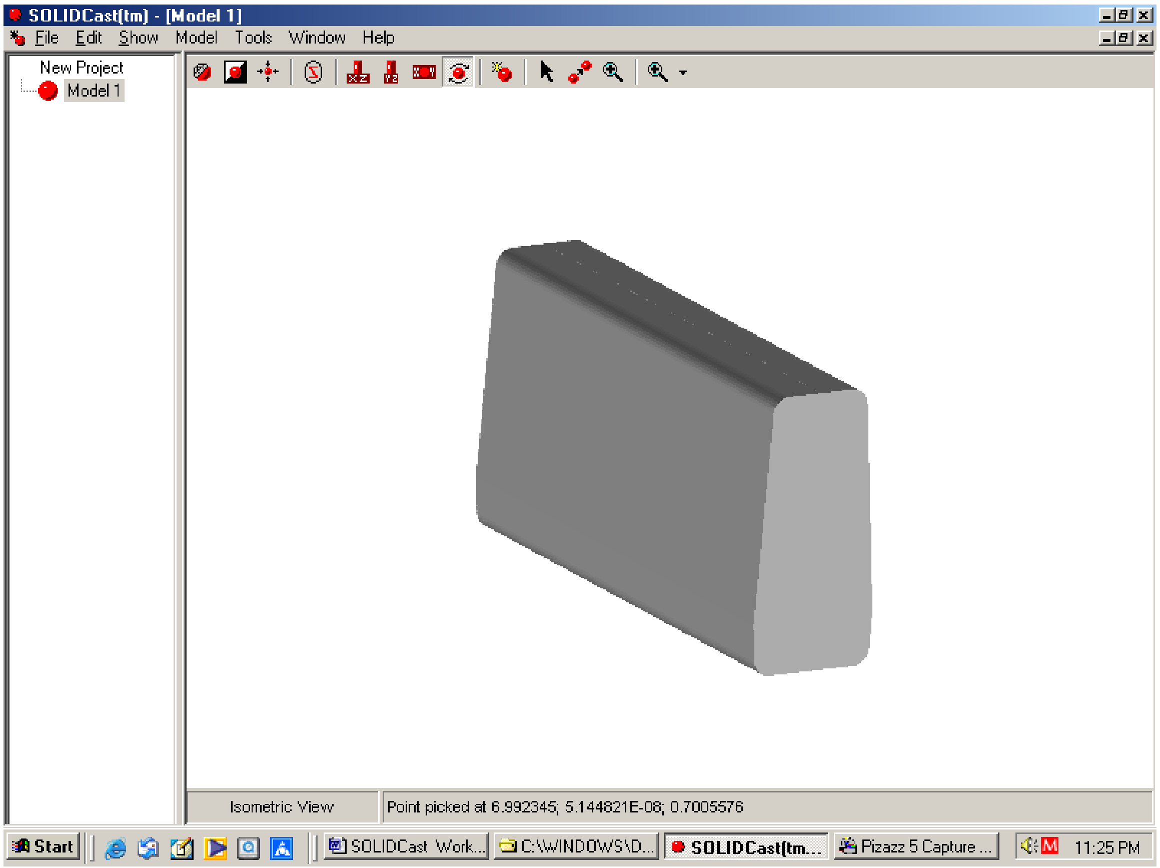
When you are satisfied that the correct shape file has been selected, and the dimensions are correctly interpreted, click on the OK button. The 2D shape will be displayed on the screen along with the Add Shape window as shown here:



You can now go ahead and fill in the remainder of the data for the shape. For example, if the extruded shape begins at Y=0.0 and ends at X=8.0, then fill in the Minimum Y and Maximum Y entries as follows:



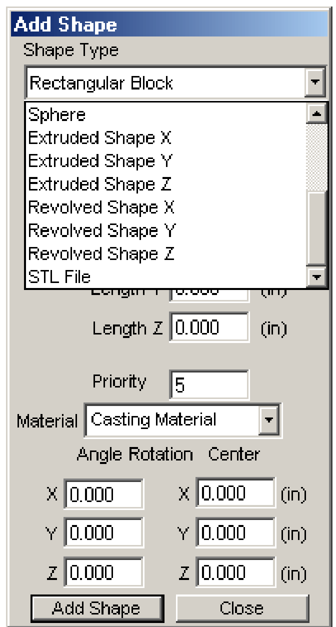
Now click on the Add Shape button. This will create the extruded shape. Note that you need to click on the Close button to get rid of the Add Shape window. In this case, after creating the shape and changing the view using the Free Rotating Isometric View, this shape would appear as follows:



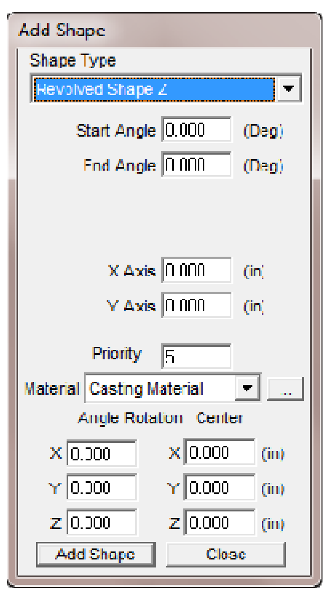
Creating Revolved Shapes from AFSCad and DXF Files

To create a solid of revolution, you must have a model displayed or a blank model space showing (select Model…New Model from the main menu to create a blank model space). Then click on the icon that says “Add a shape to the model” (you can see the icon labels if you pass the mouse over the icons on the toolbar). You will see the Add Shape window appear.

Now click on the “down arrow” and scroll down the list until the revolved shapes appear as follows:



If you select one of the revolved shapes, you will see the following appear:

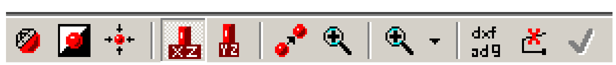


In this screen, the Revolved Shape Z has been selected. This indicates that a 2D shape will be revolved about the Z axis. This shape may be a 2D shape viewed in the XZ plane, or a 2D shape viewed in the YZ plane. The system needs to know the Starting and Ending Angles of Revolution (in degrees) and the X Axis and the Y Axis of the center of revolution. For example, a shape that is fully round will start at 0 degrees and end at 360 degrees. If the Axis of Revolution is centered on the datum point of space, then the X Axis would be 0 and the Y Axis would be 0.

When a Revolved Shape Z is selected, the system will automatically adjust the view so that you are viewing the model in the XZ view. When selecting a Revolved Shape, the system will automatically switch to one of the orthogonal views, but the tool bar at the top of the screen will be adjusted so that you can switch to one of two orthogonal views. The views that are allowed for each type of Revolved Shape are as follows:

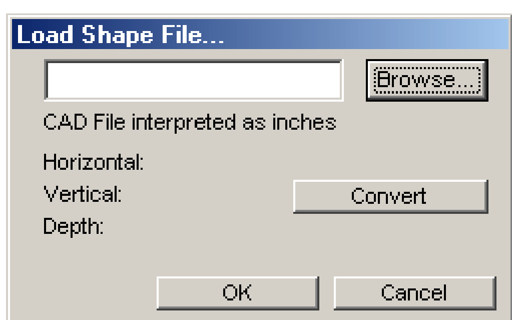
Revolved Shape X	XZ View or XY View
Revolved Shape Y	YZ View or XY View
Revolved Shape Z	XZ View or YZ View

You will notice that, when you select Revolved Shape Z, the toolbar at the top of the screen changes and appears as follows:

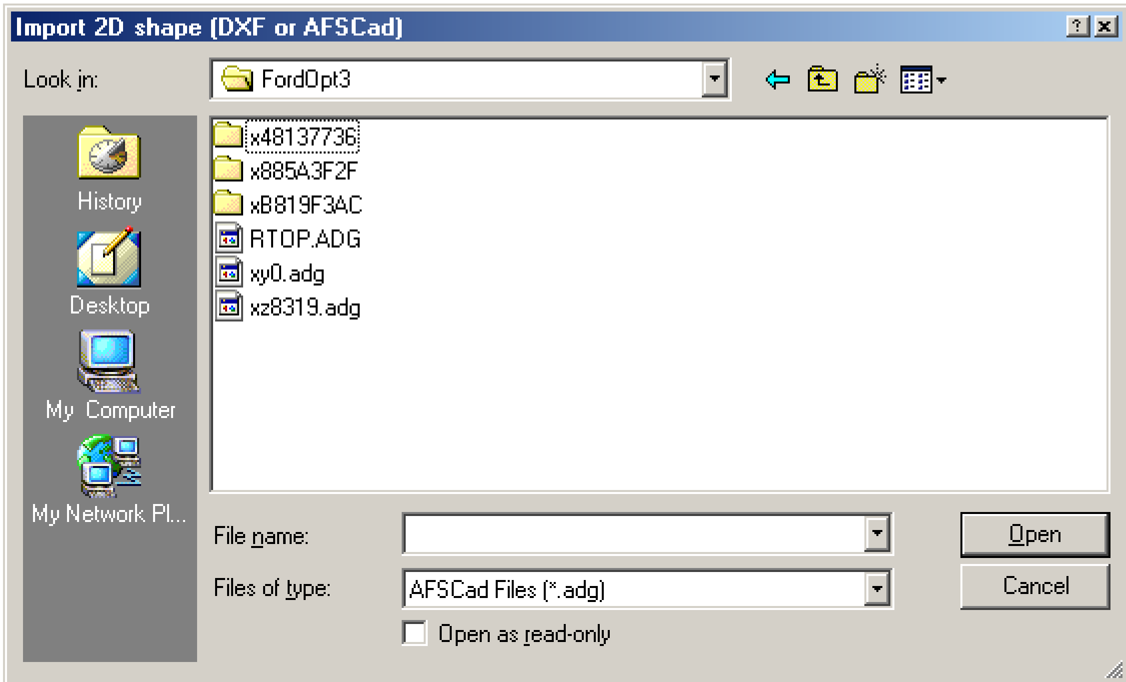


You will note that this toolbar allows you ONLY to switch between the two views XZ and XY. If the 2D shape represents a section taken in the XZ view, then stay with the XZ view. If the 2D shape is a section taken in the YZ view, then switch to this view.

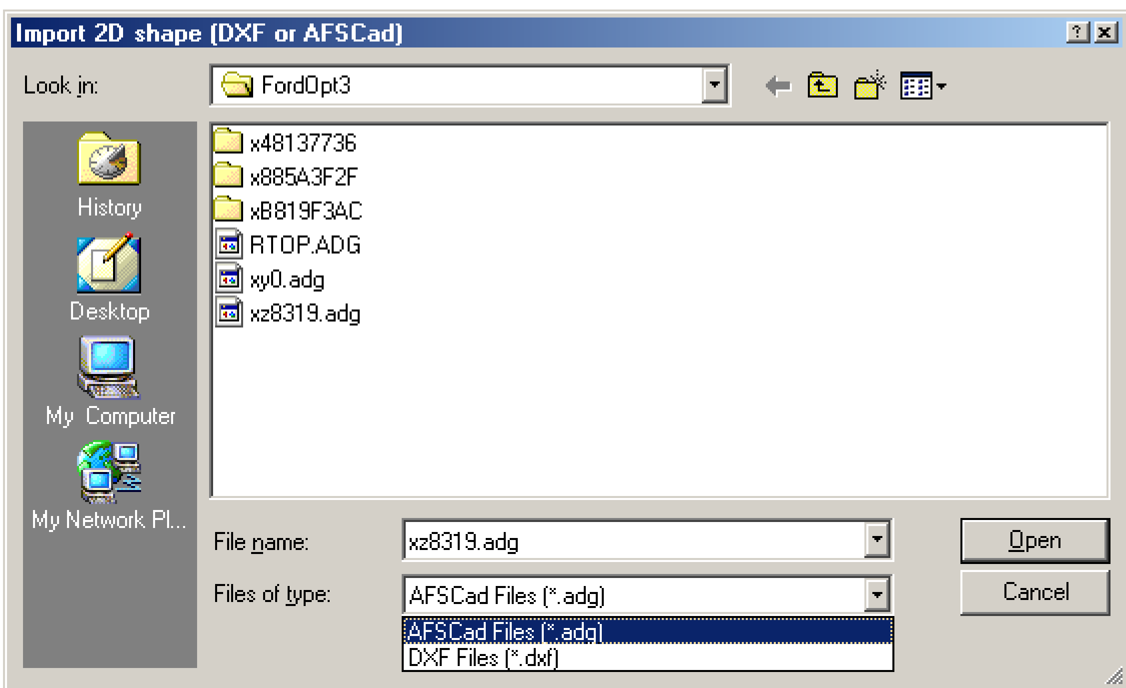
In order to indicate to the system that you will be loading a 2D file (either a DXF file or an AFSCad drawing file) click on the toolbar button labeled “dxf/adg”. You will see a window that appears as follows:



This window now allows you to select a file (DXF or AFSCad) from which to load the 2D shape. Click on the Browse button on this window. Another window will open which lets you select an AFSCad file (or a DXF file), as shown on the next page:

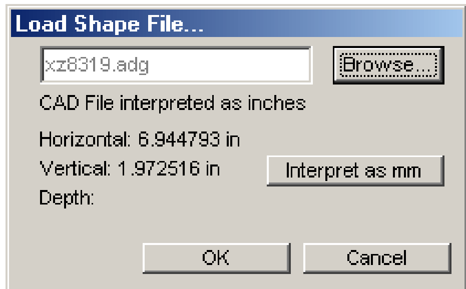


This window initially opens into the folder that is specified under System Parameters for Import Files. You can search through different folders using standard Windows navigating techniques. You can also search for DXF files by clicking on the "down arrow" next to the label "Files of type", as shown here:



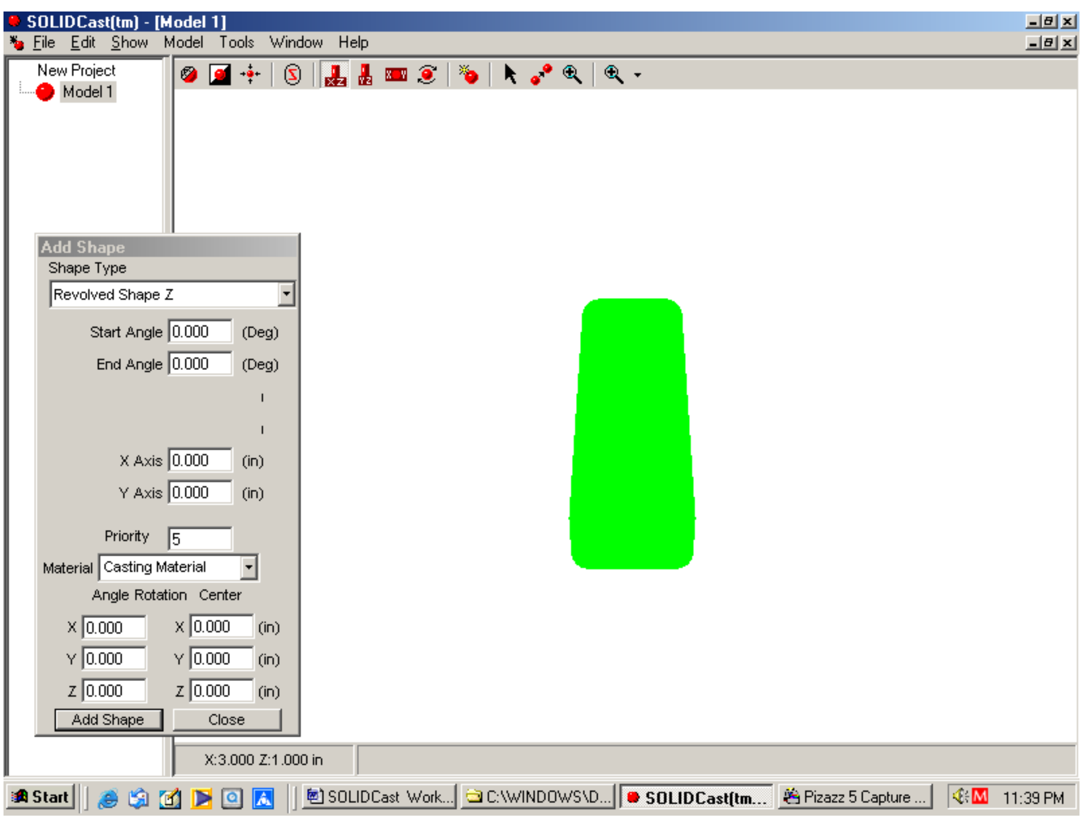
By selecting DXF Files you can then browse through any DXF files that you want to load.

After highlighting a file, click on the Open button. This will then bring you back to the preceding window, and it will appear similar to the following:

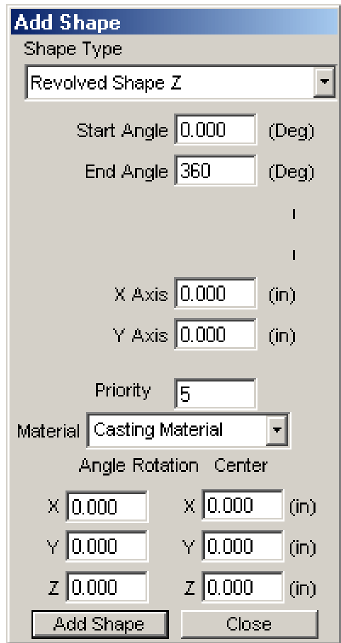


The overall dimensions of the shape (in two directions) are given on this screen. The system displays (in inches or millimeters) what it believes to be the dimensions of this shape. If the dimensions are incorrect, it may be because the shape was saved in millimeters and the system is interpreting the dimensions as inches, or vice versa. In this case, you can click on the button labeled “Interpret as mm” and the dimensions will be converted. When this occurs, the button label changes to “Interpret as inches”. Each time you click this button, it toggles back and forth between interpreting the file as inches or mm.

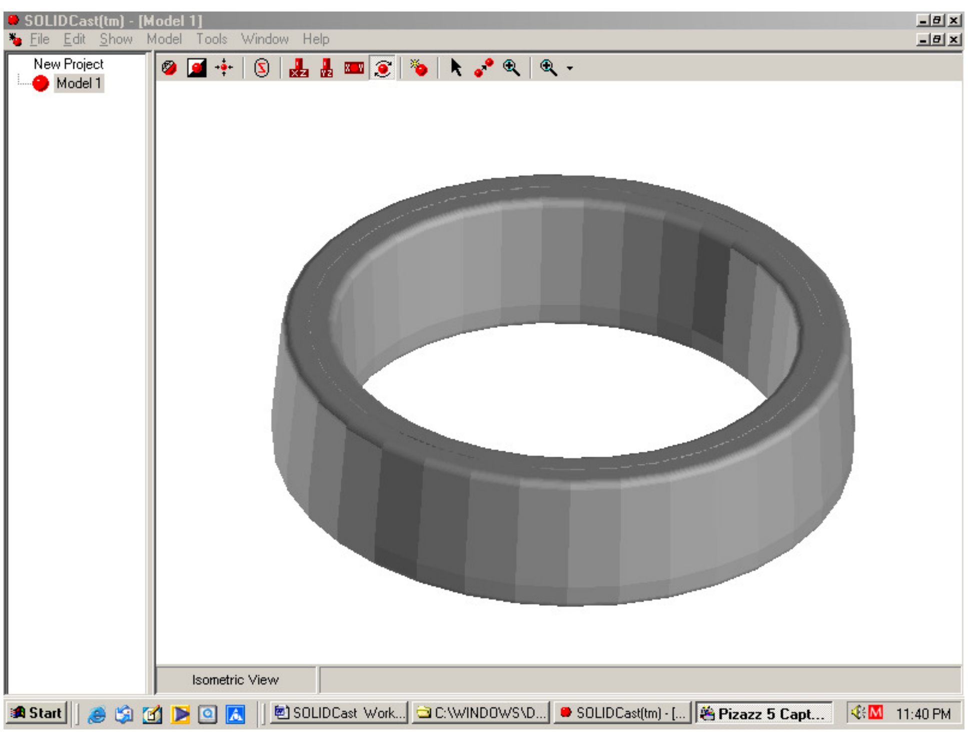
When you are satisfied that the correct shape file has been selected, and the dimensions are correctly interpreted, click on the OK button. The 2D shape will be displayed on the screen along with the Add Shape window as shown here:



You can now go ahead and fill in the remainder of the data for the shape. For example, the shape data may appear as follows:



Now click on the Add Shape button. This will create the revolved shape. Note that you need to click on the Close button to get rid of the Add Shape window. In this case, after creating the shape and changing the view using the Free Rotating Isometric View, this shape would appear as follows:

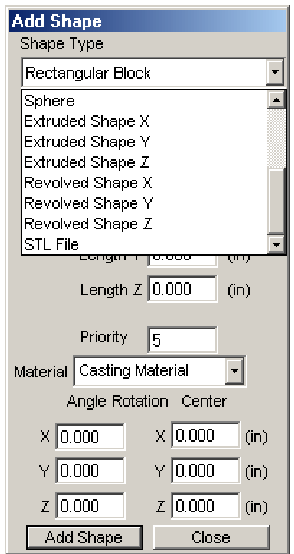


Creating Solids of Extrusion and Revolution by Sketching with the Mouse

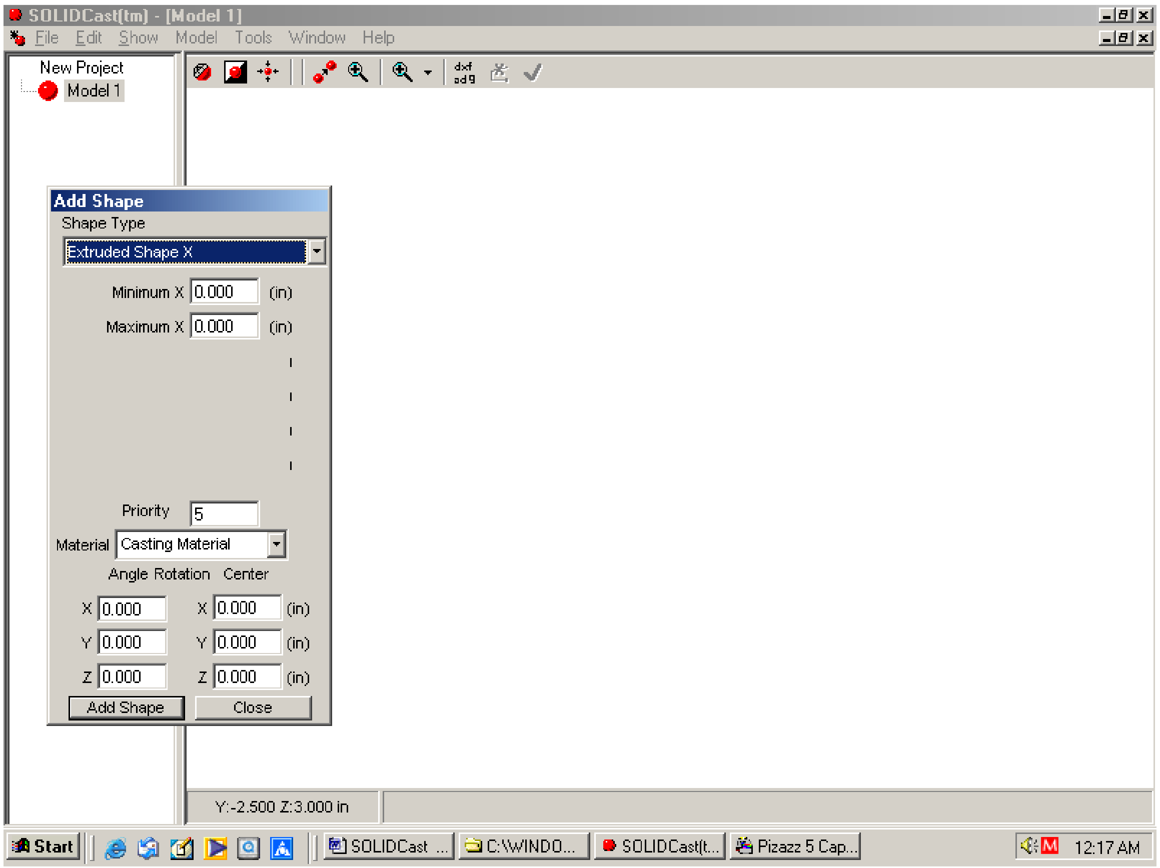
In SOLIDCast you can create a 2D shape by sketching with the mouse on the screen, and then create an extrusion or revolution with the sketched shape. You can control the mouse location more accurately if you set the Snap to Grid setting under Tools… System Parameters. For example, if you set this to 0.25 and you are working in inches, you will be able to draw to the nearest 0.25 inch on the screen.

To start the process of sketching a shape, you must first select one of the extruded or revolved solids from the Add Shape function. For example, to create an extrusion in the X direction, first click on the icon that says “Add a shape to the model” (you can see the icon labels if you pass the mouse over the icons on the toolbar). You will see the Add Shape window appear.

Now click on the “down arrow” and scroll down the list until the extruded shapes appear as shown following:



If you select Extruded Shape X, the screen will appear as follows:



Notice that, when an Extruded Shape X is selected, the system will automatically adjust the view so that you are viewing the model in the YZ view. When selecting Extruded Shape, the view is automatically adjusted as follows:

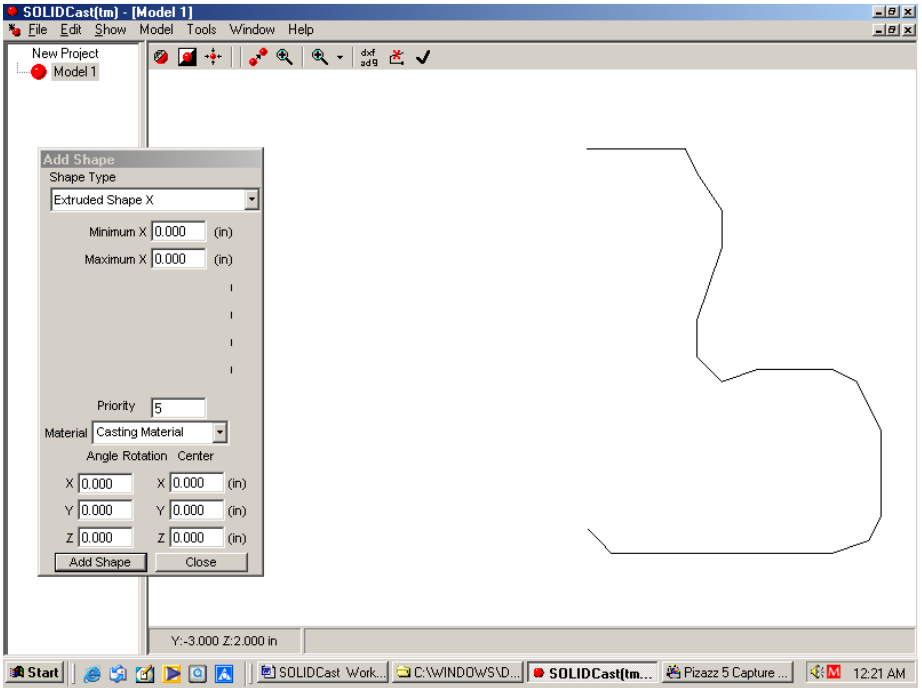
Extruded Shape X	YZ View
Extruded Shape Y	XZ View
Extruded Shape Z	XY View

Now you can begin drawing with the mouse, in the clear model space to the right of the Add Shape window (you can drag the Add Shape window to the side if it is in the way). Notice that the mouse coordinates (Y and Z) appear at the bottom of the screen. To begin drawing a shape, just click with the mouse.

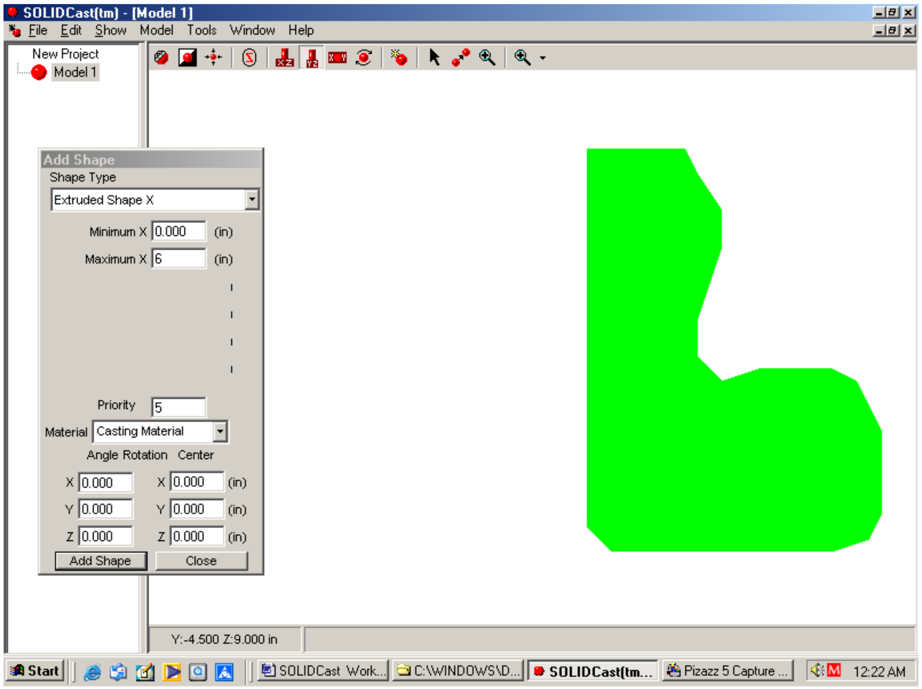
Once you start clicking, you will notice that the toolbar at the top of the screen has changed again. While you are sketching, it appears as follows:

The icon with the "Check Mark" indicates that you are finished sketching the shape. The icon to the left of the check mark is used to delete the previous line drawn.

When sketching a shape, you are actually drawing a series of straight lines on the screen. The starting point is the first point where you click. As you click on additional points, a series of lines will be drawn. The sketching process may appear as follows:

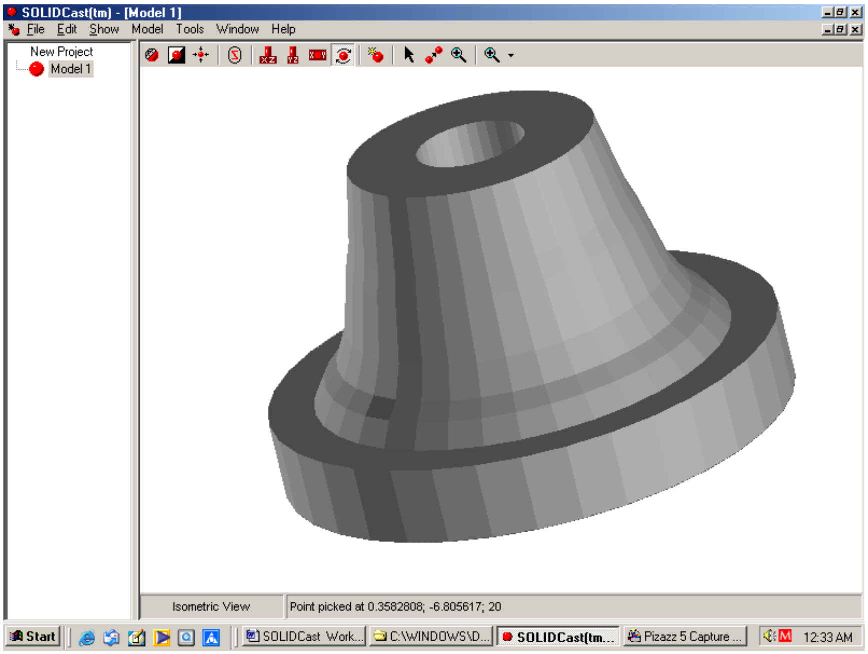


You do NOT need to draw the last line to close the figure. When you finish sketching, click on the Check Mark icon at the top of the screen. This will close the last line drawn back to the starting point, and the screen will then appear as follows:

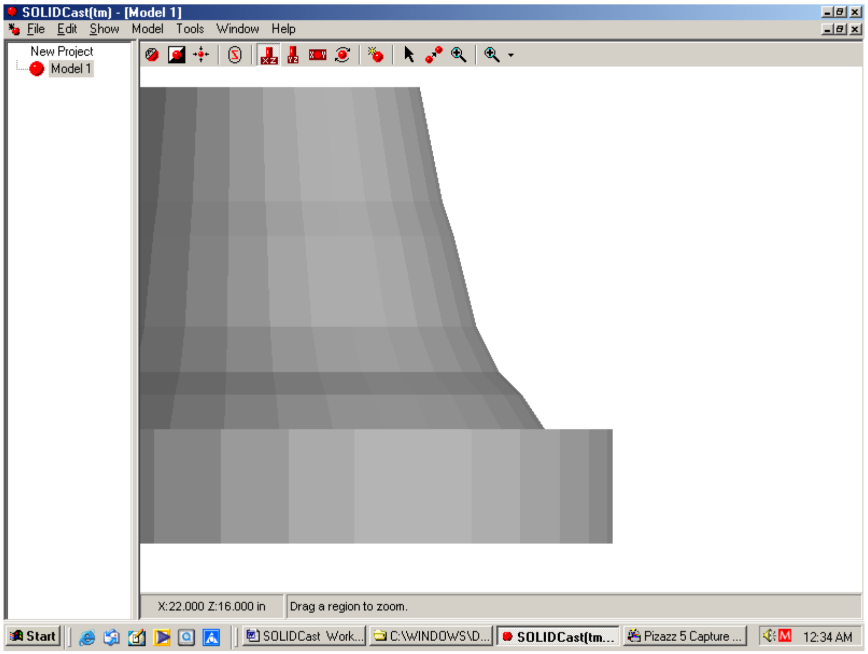


At this point, you need only to fill in the shape parameter data and click on the Add Shape button. This will create the 3D extruded shape in the model space.

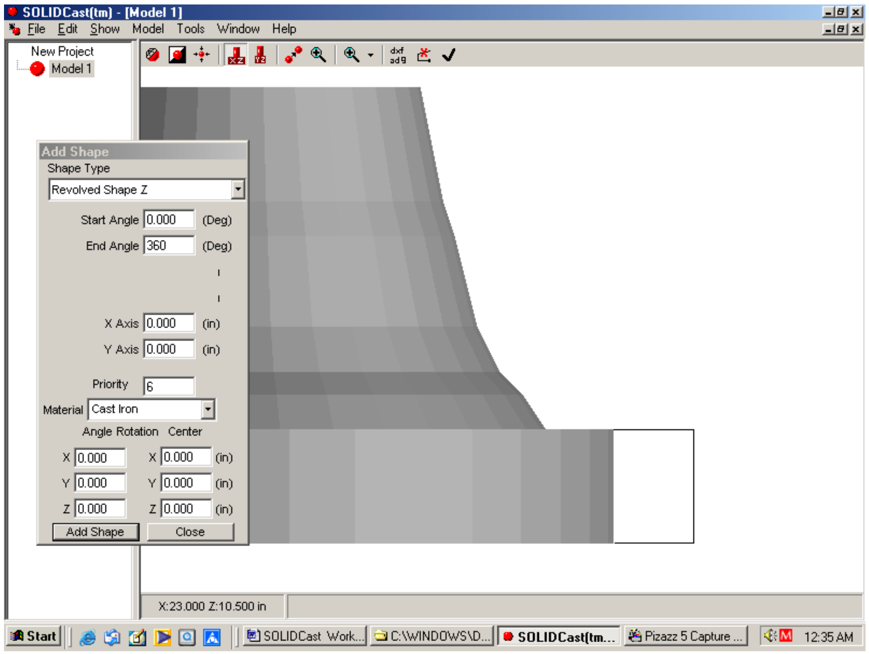
One fairly common use for "shape sketching" is to place chills on a casting model. For example, suppose we wanted to add a series of cast-to-shape curved chills around the outer perimeter of the following casting (this is a round casting that is centered on (0,0)):



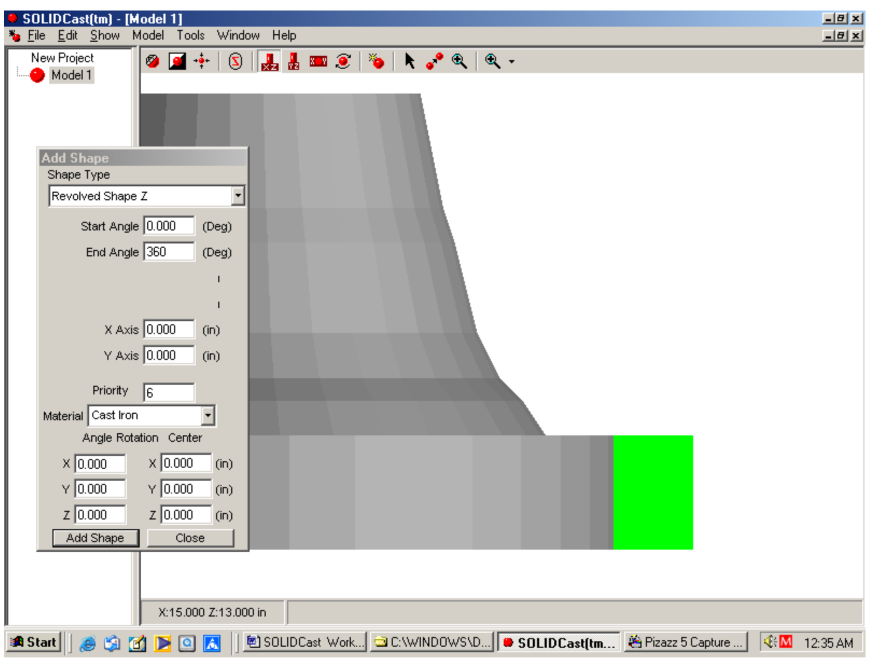
We could start by changing to the XZ view and then zoom on a portion of the casting in order to see that portion in more detail:



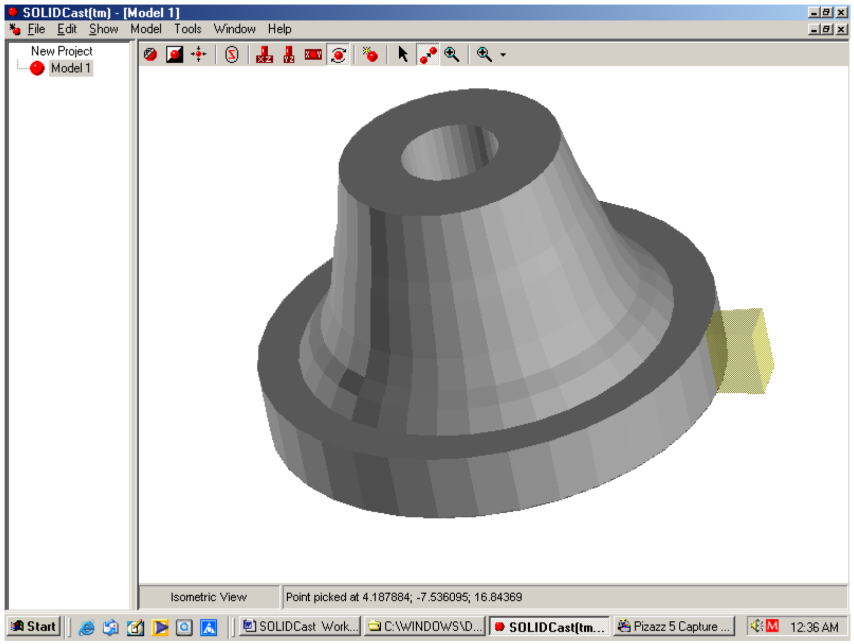
We would then select Add Shape, and select Revolved Shape Z. Assuming that we had the Snap to Grid setting at 0.25 inches so that we could draw accurately, we could then draw a rectangular cross section of a chill as shown in the following picture:



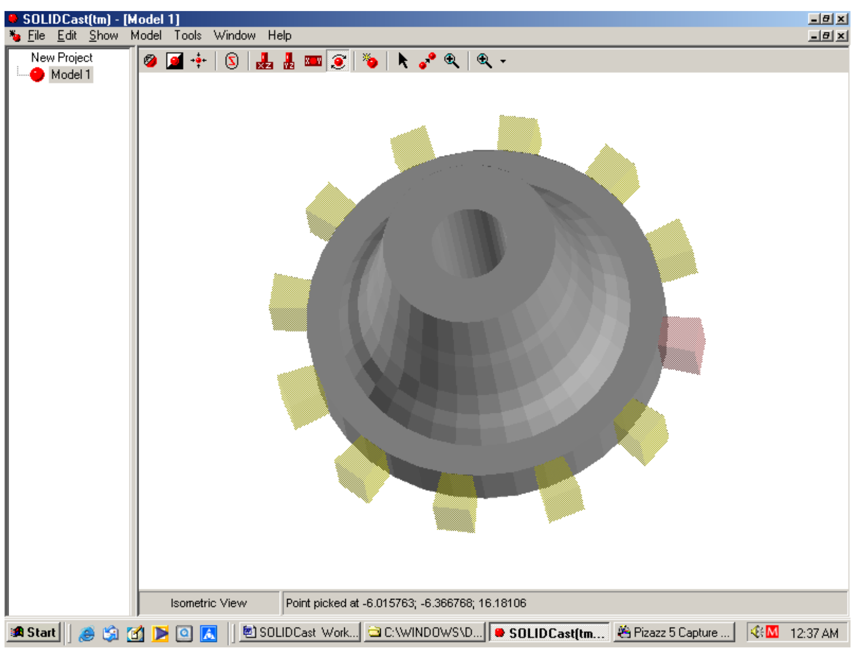
Clicking on the Check Mark icon finishes the shape, as follows:



Selecting a starting angle of –5 degrees and an ending angle of 5 degrees defines a chill that extends 10 degrees around the circumference. We also select a Chill Material (say, Cast Iron) for this shape. Once these selections have been made, the chill appears as follows:



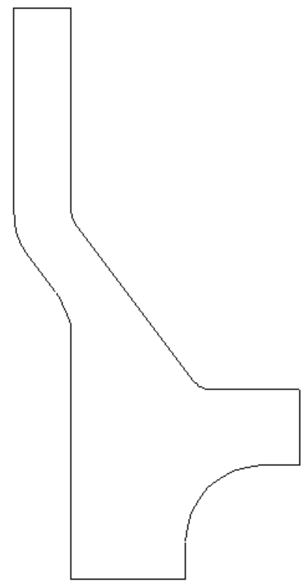
Now it is possible to Edit this one shape, perform a Ring Copy around the point (0,0) and make a copy, say, every 30 degrees around the casting. The end result of this operation would appear as follows:



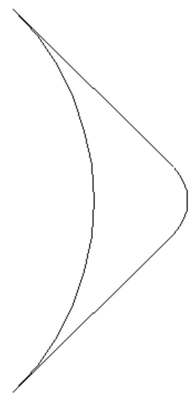
We now have 12 chills equally spaced around the perimeter of the casting.

As an example of creating a casting shape using solids of revolution and extrusion, consider the following 2D figures that were created in a CAD system and saved in DXF format, which is available in virtually any 2D CAD system today:

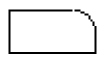
Saved in file Y-1.DXF:



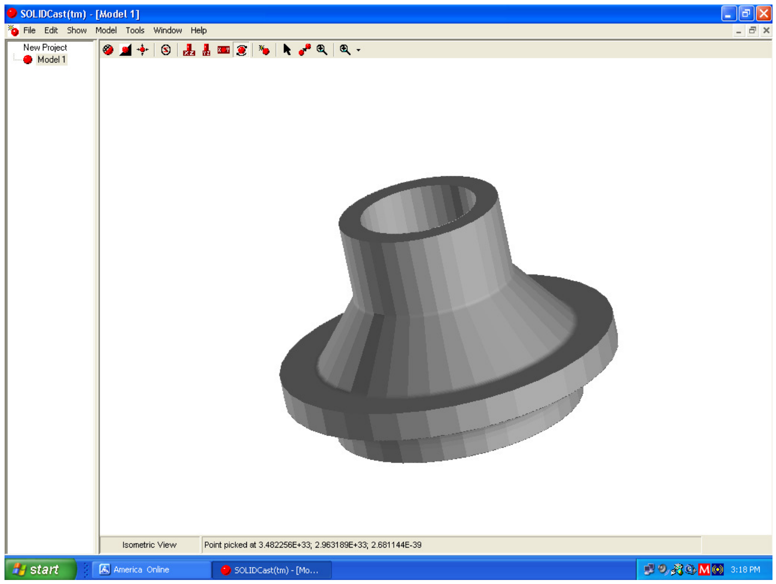
Saved in file Y-2.DXF:



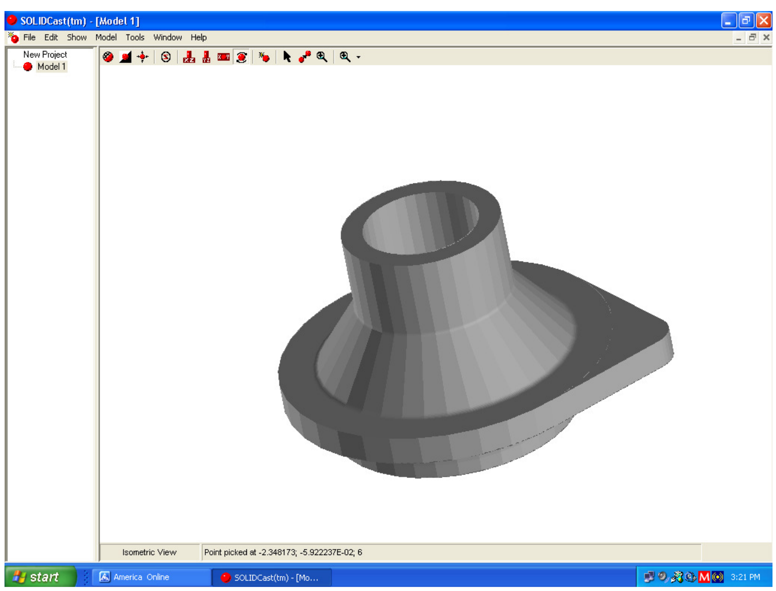
Saved in file Y-3.DXF:



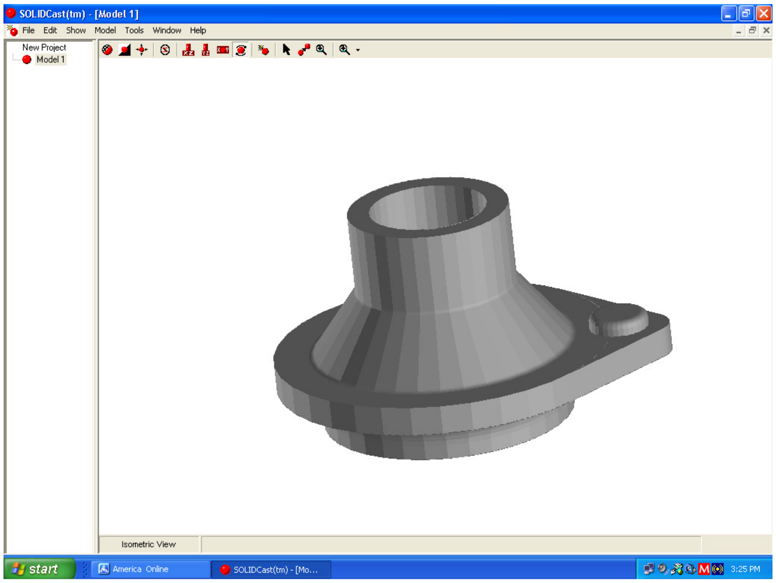
The process of creating a casting model from these shapes would be as follows; First, select File…New Model. Then select Add a shape to the model, select Revolved Shape Z, select the DXF file called Y-1.DXF, specify a revolution of 0 to 360 degrees, leave the center of revolution at X=0, Y=0, and then click on Add Shape. The first shape will appear as follows:



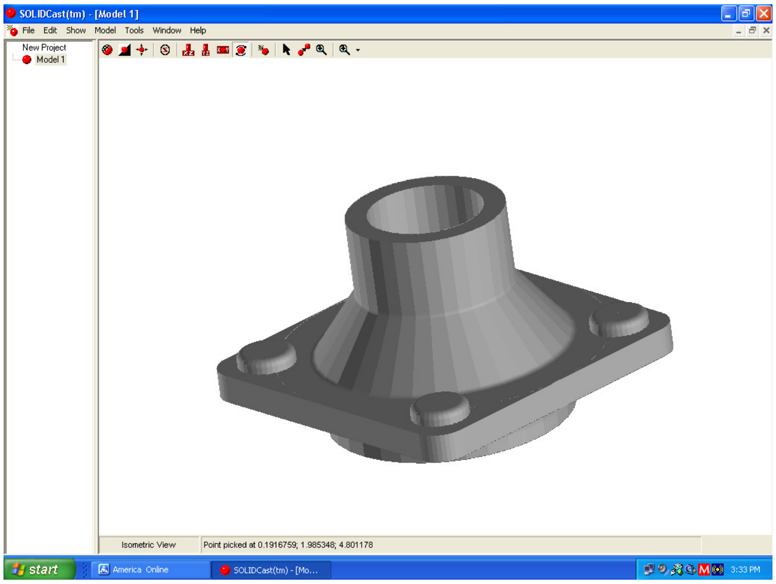
Now we will add an extrusion to this shape. Select Add a shape to the model, select Extruded Shape Z, select the DXF file called Y-2.DXF, set the extrusion from 0 to 1 inch, and click on Add Shape. The second shape will appear as follows:



Now add the third shape as a revolved shape. To do this, select Add a shape to the model, select Revolved Shape Z, select the DXF file called Y-3.DXF, specify a revolution of 0 to 360 degrees, set the X coordinate of the center of revolution to 6.000 inches, and then click on Add Shape. The third shape will appear as shown here:

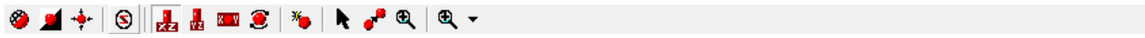


Now we can duplicate the second and third shape by a Copy operation (see the section of Editing/Copying for details). Select both the second and third shapes (by clicking the "Selection" arrow icon, holding down on Ctrl and clicking both of these shapes) then select Edit… Copy Selected Shape(s) and specify a Ring Copy about the Z axis with (0,0) at the center, with three copies at 90 degrees. This will produce the following final model for this casting:



UNIT 12: Display Controls

In SOLIDCast, the display controls are located as icons across the top of the model window. These display controls appear in a bar as follows:



The meanings of these icons are as follows:

 Switches between a shaded rendered view and a wire-frame view of the casting.

 Toggles the background between black and white.

 Centers the model in the current window.

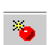
 Reverses the current 2D view. From top to bottom, front to back, or side to side.

 Displays an XZ view of the casting model (this is the default view when you load a file).

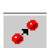
 Displays a YZ view of the casting model.

 Displays an XY view (looking down) of the casting model.

 Allows the user to freely rotate the casting model to any orientation, by using the mouse with the right button depressed.

 Add a shape to the casting model.

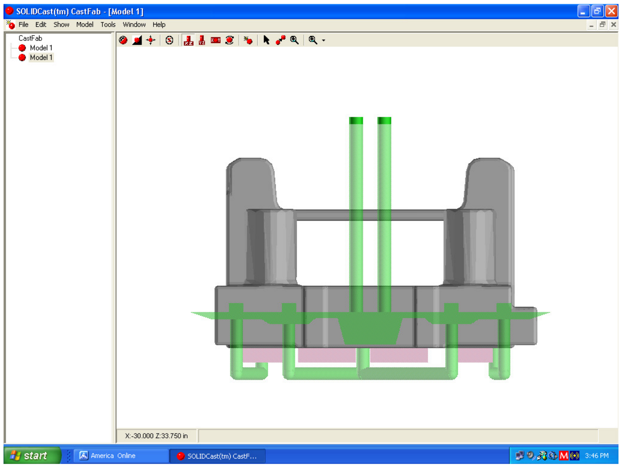
 Select a shape to hide or delete.

 Move the model on the screen by clicking on two points.

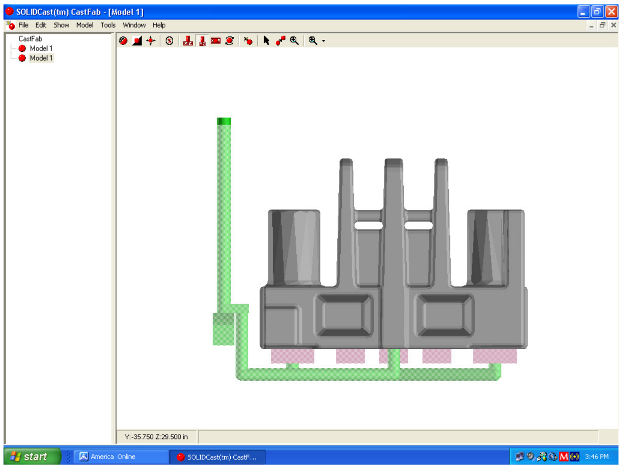
 Zoom on a portion of the model by drawing a box around the portion to be zoomed.

 Zoom up or down a fixed percentage.

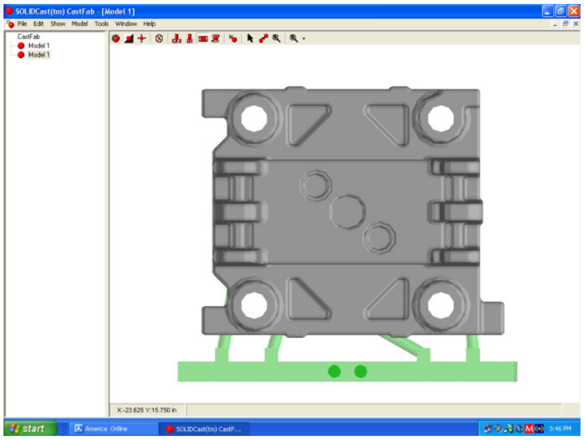
Example Orthogonal Views:



XZ View



YZ View

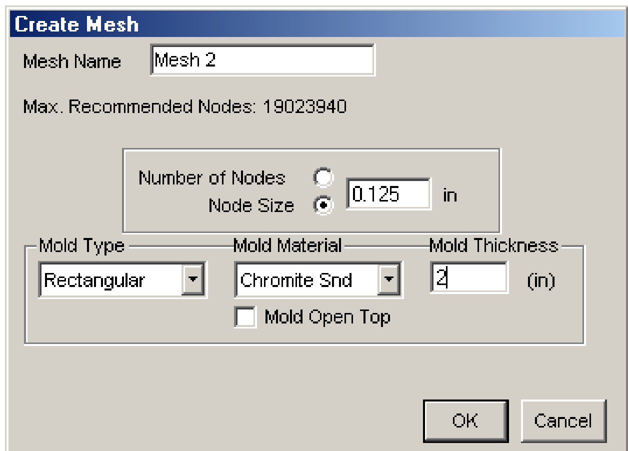


XY View

UNIT 13: Generating a Mesh

Once you have a model loaded and materials selected, the next step before running a simulation is to generate a mesh. This is done by selecting Model… Create Mesh… from the menu bar.

The system will display the meshing screen. This is where you describe how the mesh is to be generated, and it may appear as follows:



Here you can enter a descriptive name for the mesh. You can also select either the Number of Nodes (elements) or the size of the Nodes (in inches or mm). Generally, most simulations are run with between 500,000 and 20,000,000 elements. A "chunky" casting will generally require fewer elements, while a thin-section casting will require more elements. The maximum number of elements allowed depends on the amount of installed RAM.

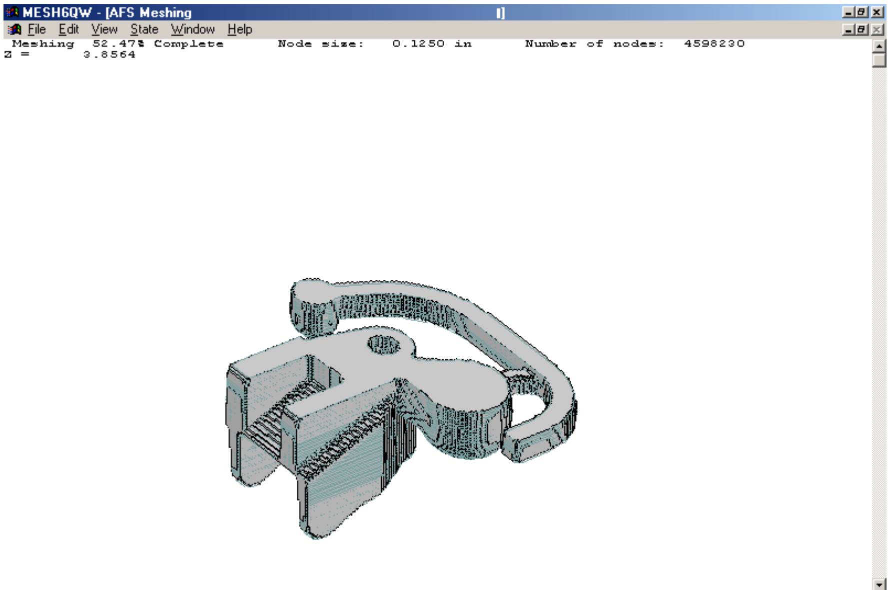
The system will automatically create a mold around the casting if you want it to. (Note: You can create the mold as part of the model, in which case you would NOT use the automatic mold creation feature.) You have three options here:

Rectangular
Shell
None

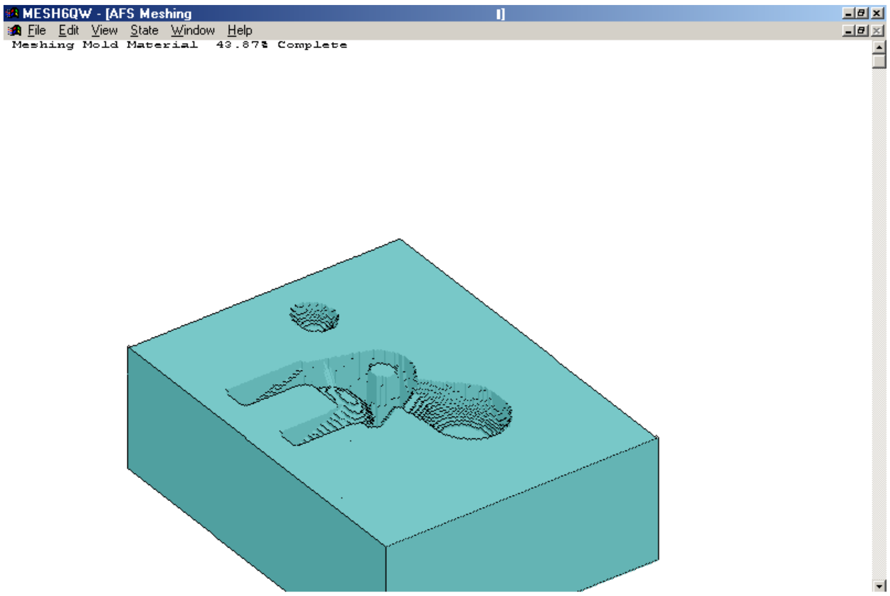
If you are going to have the system create a mold, you can select the Mold Material, the Mold Thickness (this is the minimum thickness in the case of a rectangular mold, or the shell thickness in the case of a shell mold) and also whether or not the top of the mold is to be considered open.

After making these selections, click on OK.

The system will first mesh the casting model and display the mesh as it is being created, as seen here:



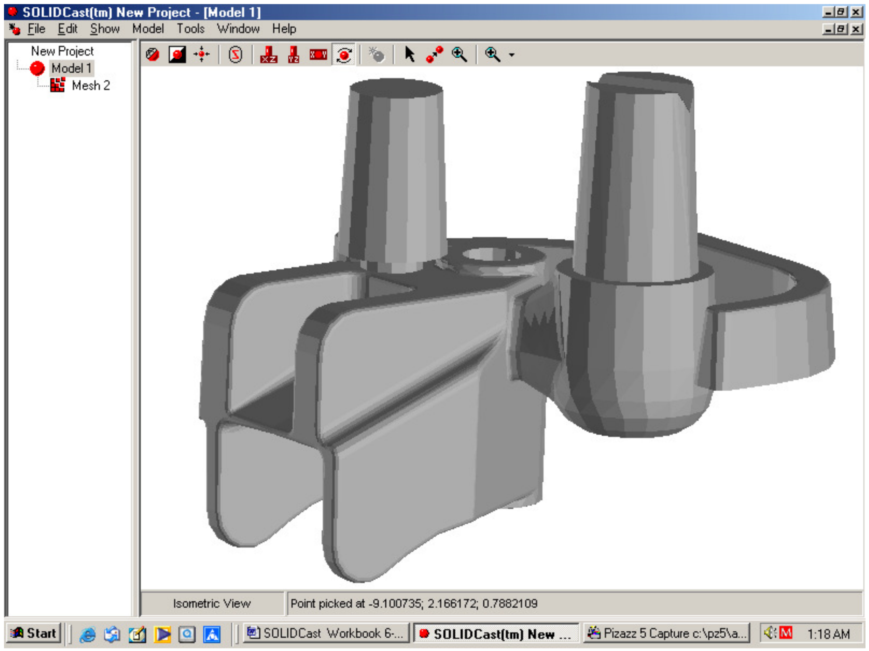
Next, the system will mesh the mold around the casting, as shown below (in this case, a rectangular mold):



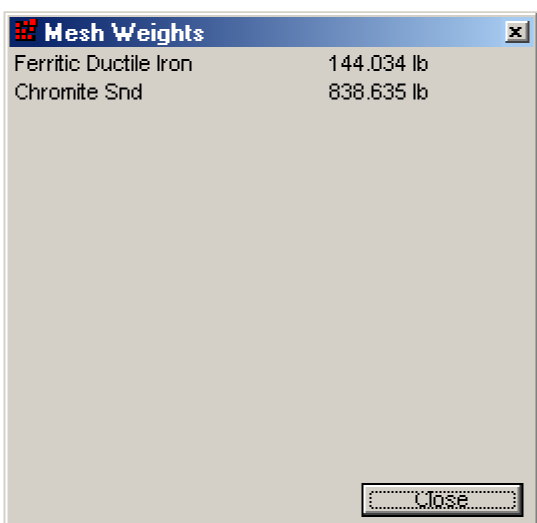
When the meshing process is complete, the mesh name will appear on the project tree on the SOLIDCast main screen.

UNIT 14: Weight Calculations

To calculate weights, you must have previously meshed a model. The mesh name appears on the project tree on the left side of the screen, as shown below:



To perform a weight calculation, highlight the mesh name and select Mesh…Weights. The following window will appear:



Note: To obtain a highly accurate weight estimate for a casting, you can create a mesh with no mold material and a high number of nodes. This may often be the first step in developing a cost estimate for a casting. Once you have gotten the weight information, you can delete the mesh to save space on the hard drive.

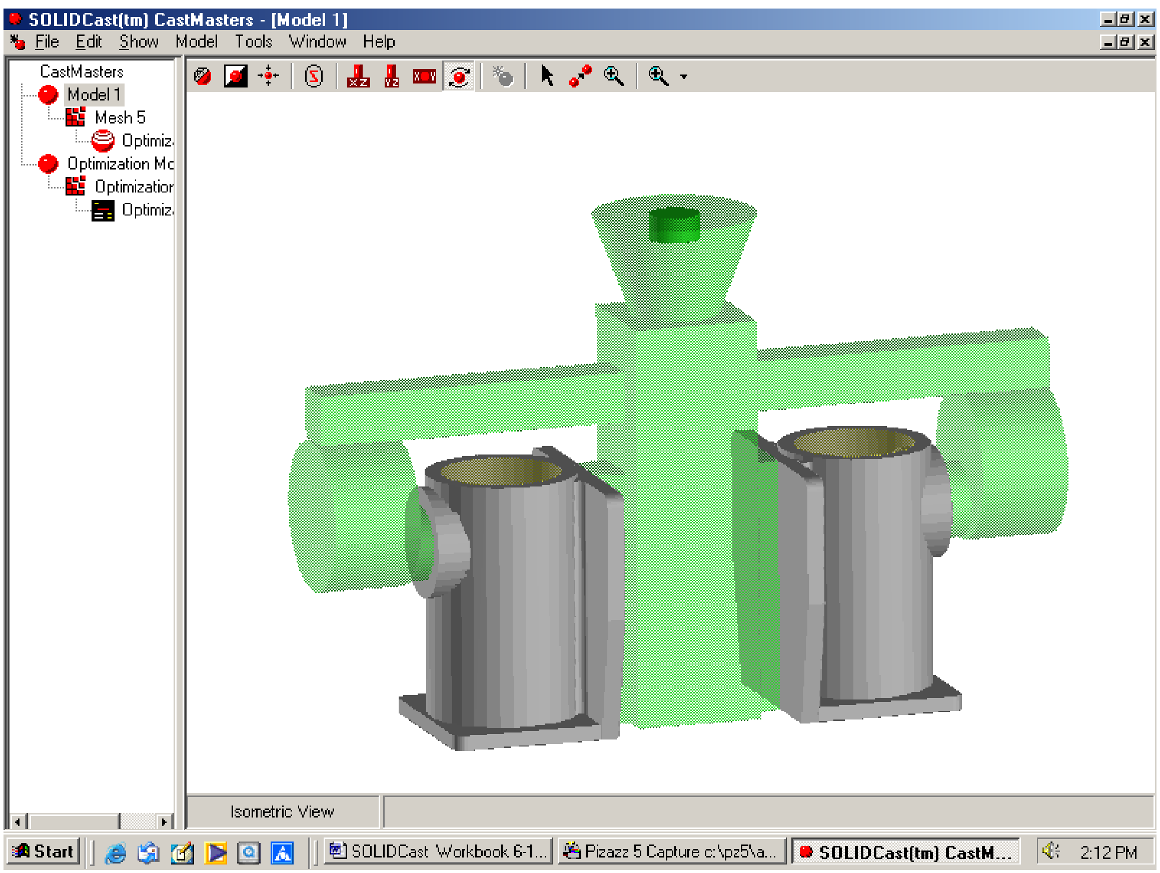
UNIT 15: View Factor Calculations

When castings are made by the investment casting or permanent mold process, a considerable amount of heat is lost from the surface of the hot shell or die in the form of radiation. The amount of radiant heat loss varies according to whether various surfaces face the ambient surroundings or other hot shell or die surfaces. When facing other shell or die surfaces, the temperature difference is smaller and the amount of radiant heat loss is less.

This variation in radiant heat loss can be simulated by a process of applying “View Factor” calculations to the mesh. The View Factor Calculation takes into account the visibility of all shell or die surfaces to all other shell or die surfaces as well as the surrounding environment, and adjusts the conditions at each surface accordingly.

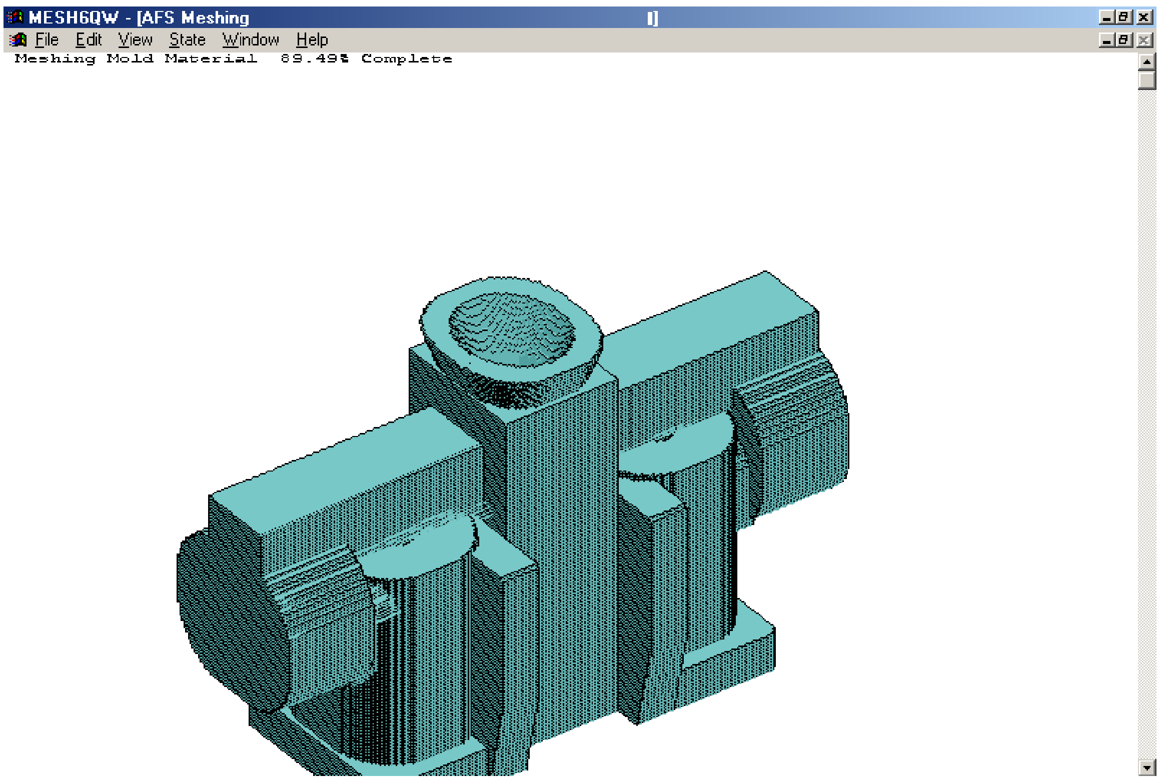
View Factor Calculations are applied to a mesh, AFTER a model has been meshed.

As an example, consider the following model of an investment casting with two castings on a tree:



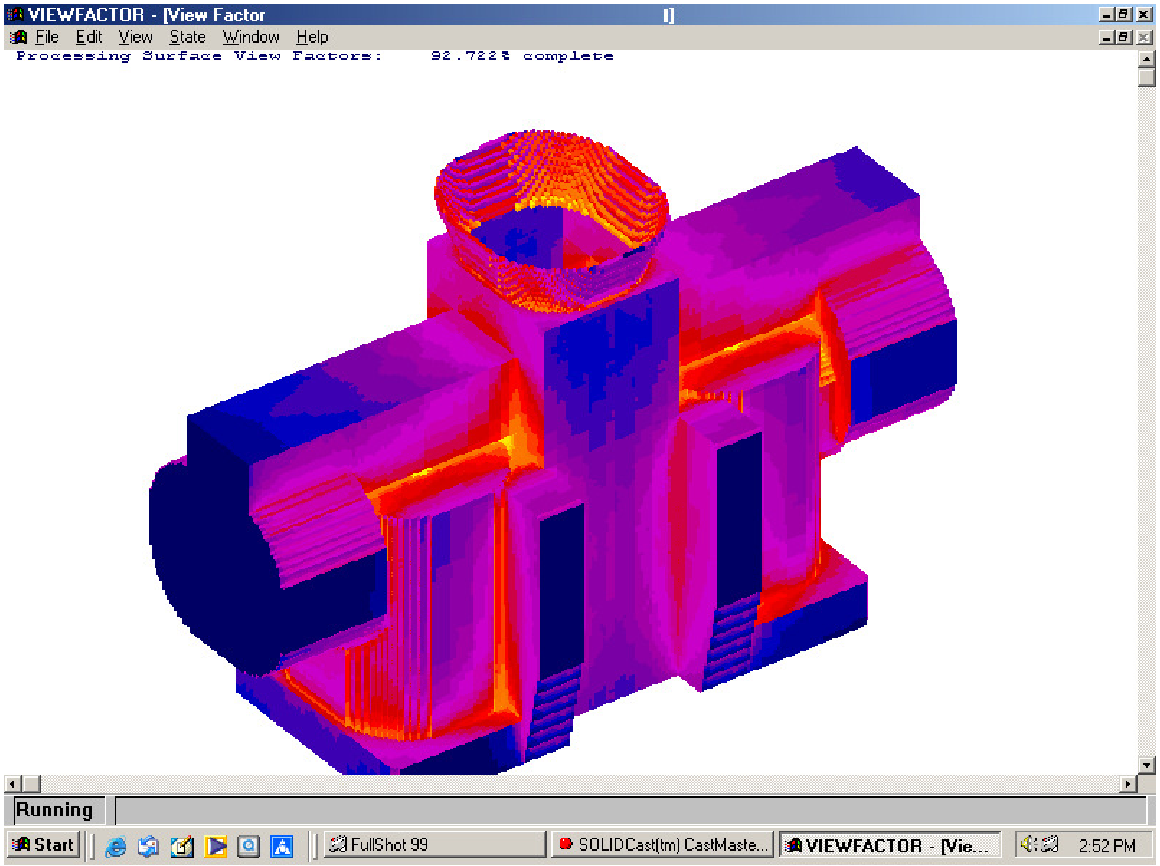
The first step in the View Factor Calculation is to select Model…Materials List and click on the HT Coefficients tab. The value that you place in the External HT Coefficient box will be the 'high' value used for radiation heat transfer. This number is related to the temperature of the shell or die, and is typically about 5-7 BTU/hr-sq ft-F for permanent mold dies and 10-20 for investment shells. Values for use can be calculated using the HTC Calculator utility.

Once the External HTC is set, mesh the model by clicking on Model…Create Mesh. Meshing this example casting with a shell mold results in the following investment shell:



Now, having created the shell by meshing, we can apply the View Factor Calculation to take into account the variation in radiation heat exchange around the surface of the shell.

To do this, we first click on the Mesh icon on the project tree to highlight the mesh. Then, on the menu bar at the top of the window, select Mesh and then View Factor Calculation. The calculation will be performed and displayed as shown on the next page:



In this view, the dark areas on the shell are losing heat most rapidly due to a high rate of radiant heat exchange with the surroundings. The lighter areas are those that see mainly other portions of the hot shell, and those areas are losing heat more slowly as they exchange radiant heat with those other portions of the shell.

After having performed the View Factor Calculation, you can go on to the next step and run a simulation. The View Factor adjustments are now built into the mesh.

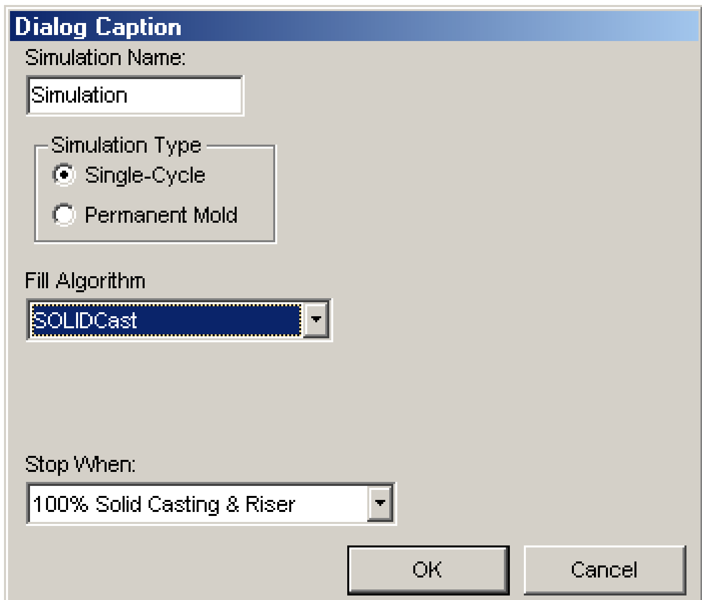
By the way, multiple View Factor Calculations will not change the mesh. So, if you can't remember whether you did the calculation or not, select it again to be on the safe side.

In the same way, the View Factor Calculation can be applied to a permanent mold casting. The view factors are applied to every surface in contact with ambient conditions, so it doesn't matter if the die/shell is created as a part of the model, or by meshing.

In general, View Factor Calculations are of limited use in sand casting simulations and would not be applied to such a model, due to the low temperatures on the outside of the mold during solidification. A possible exception would be the shell molding process, where the mold thickness can be quite small compared to part thickness.

UNIT 16: Running a Simulation

To run a simulation, you must have previously created a mesh. The mesh name will appear on the project tree on the left side of the SOLIDCast main screen. Highlight this mesh name, then from the menu bar select Mesh… Start Simulation. The following window will appear:



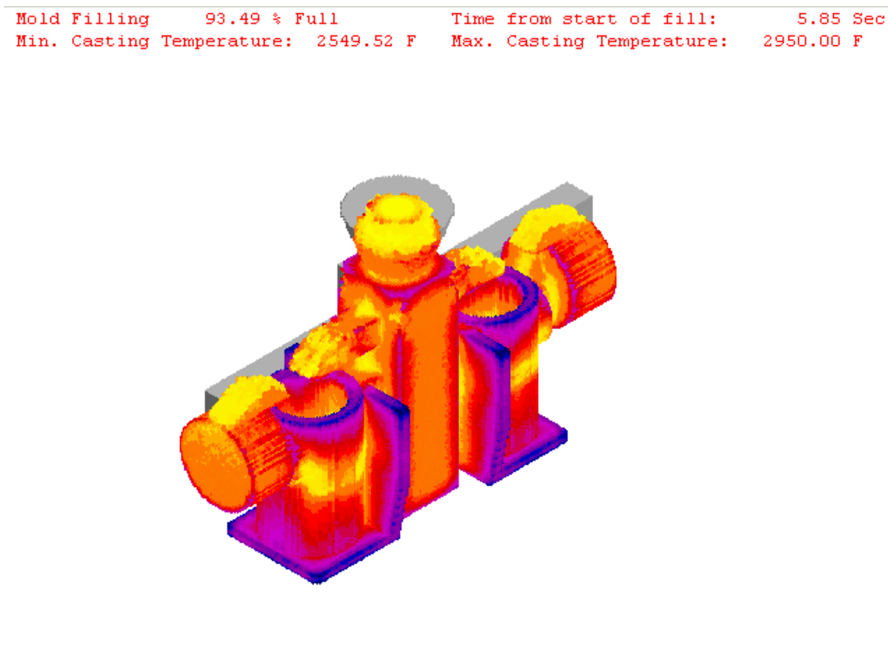
You can select to run either a single cycle (such as a sand or investment casting) or a multiple-cycle permanent mold simulation.

If you have added Fill Material to a model and if you have the FLOWCast Fluid Flow module installed, you will have a box labeled Fill Algorithm, and will have the choice of using the simple SOLIDCast fill algorithm, or the FLOWCast Quick or FLOWCast Full algorithms.

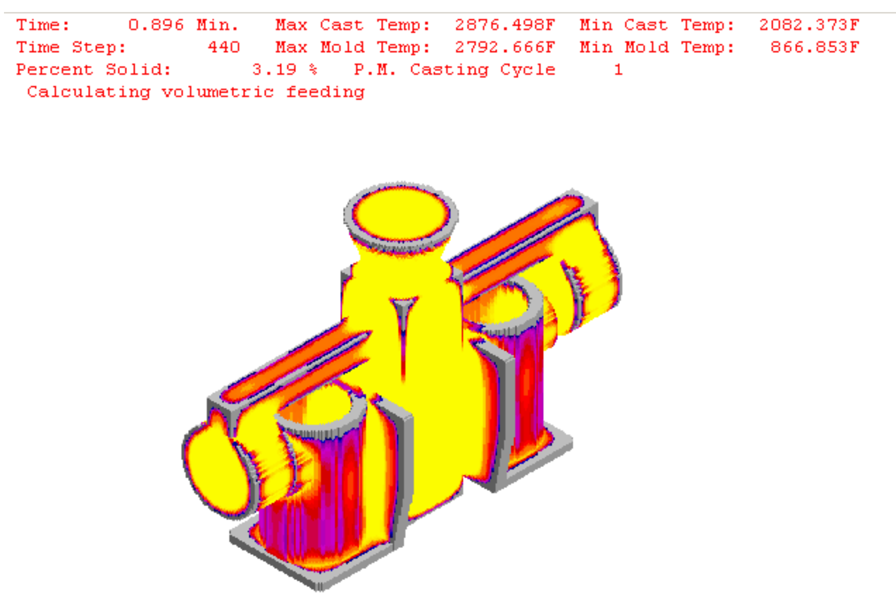
You can also select the criterion that the system uses to stop the simulation and consider it to be complete. The most common stop criterion is for the system to end the simulation when the casting and risers are 100% solid.

To start the simulation running, click on OK.

If you selected the SOLIDCast fill algorithm, you will see a graphic picture of the casting filling while the simulation is running, such as in the following image:



Once filling is complete, the display will switch over to a summary screen, showing the relative temperatures in the model and other information. If you have unchecked the box in System Parameters that controls the graphic display during simulation, you will see a text screen which lists a summary of simulation progress. A sample simulation screen is shown here:



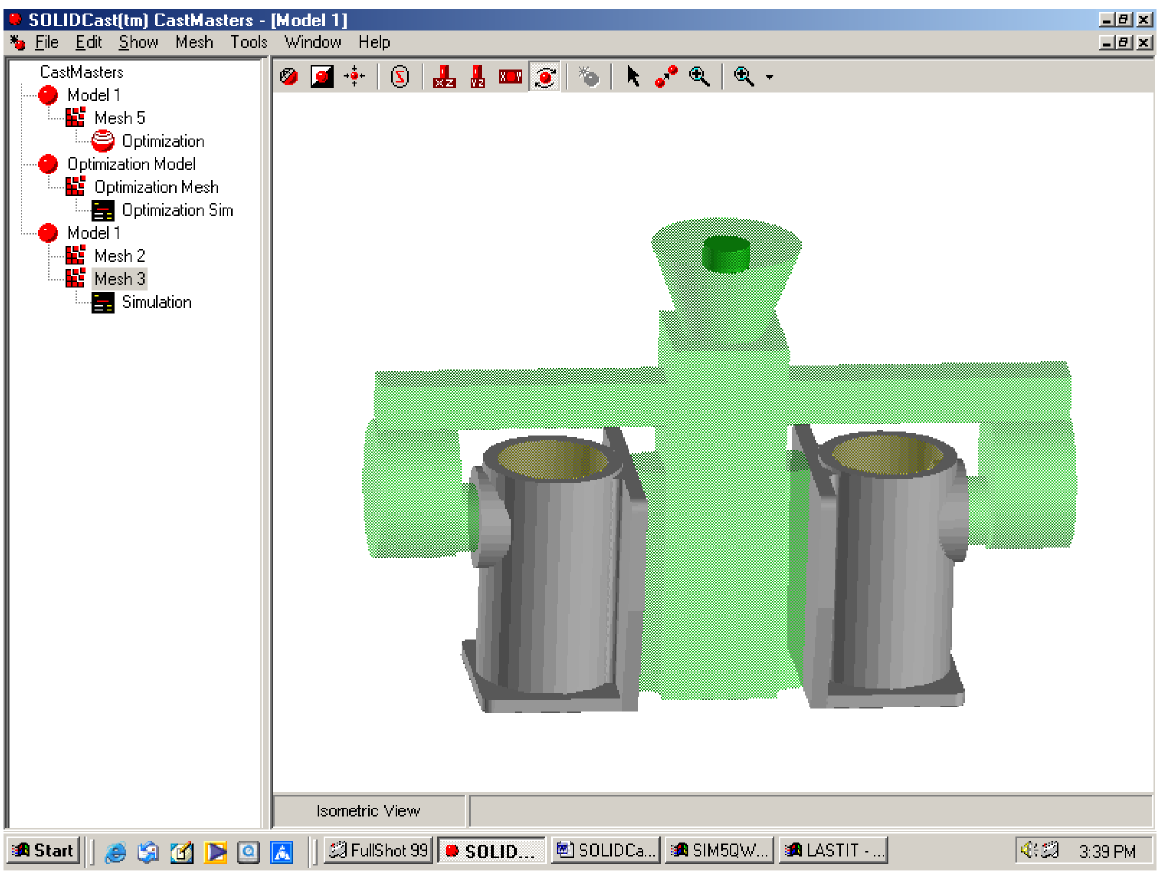
You can minimize the windows associated with the simulation and use the computer for other programs such as word processing while the simulation is running.

Once the simulation is complete, you are ready to start plotting results.

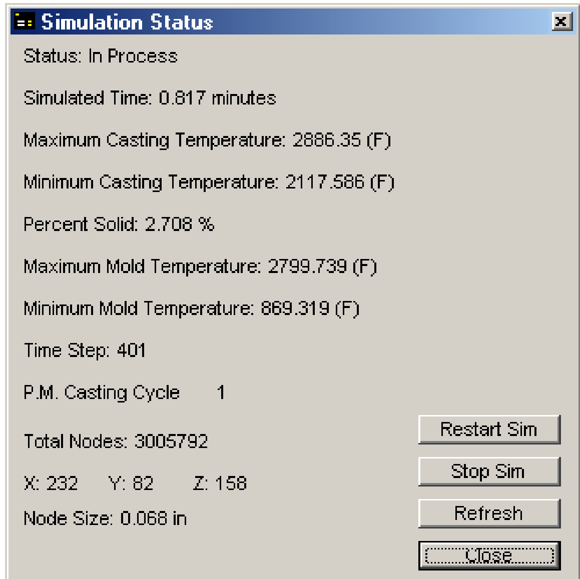
Stopping and Restarting Simulations

In order to stop a simulation such that partial results can be plotted, or so that the simulation can later be restarted from the same point and completed, it is necessary to view the project tree that the simulation is running.

When you run a simulation, you will notice that two windows appear. One window is titled LASTIT (this is the window pictured on the previous page) and the other is titled SIM5QW. In order to view the project tree, you need to minimize both of these windows. This can be done by clicking on the “Minimize” button (the button with a short horizontal line) in the upper right corner of each of these windows. This will minimize these windows so that they appear on the Windows Task Bar, and the SOLIDCast window with the model and the project tree will appear on the screen. This may appear as follows:

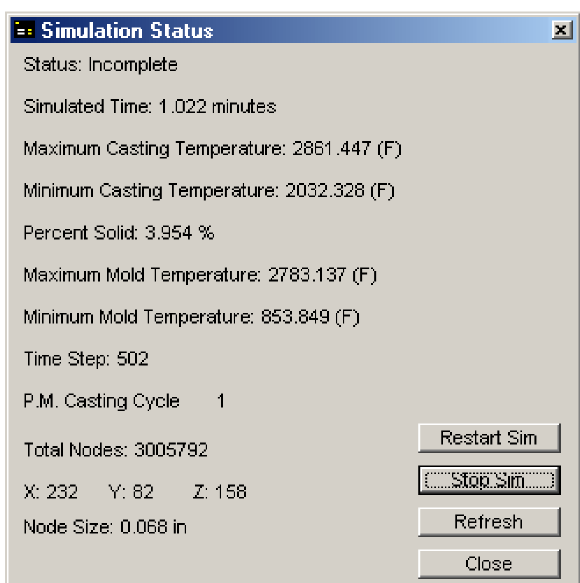


On the project tree, which appears on the left side of this screen, you will see an entry for this simulation. To stop the simulation, double-click on the icon next to the simulation on the project tree. A window similar to the following will appear:



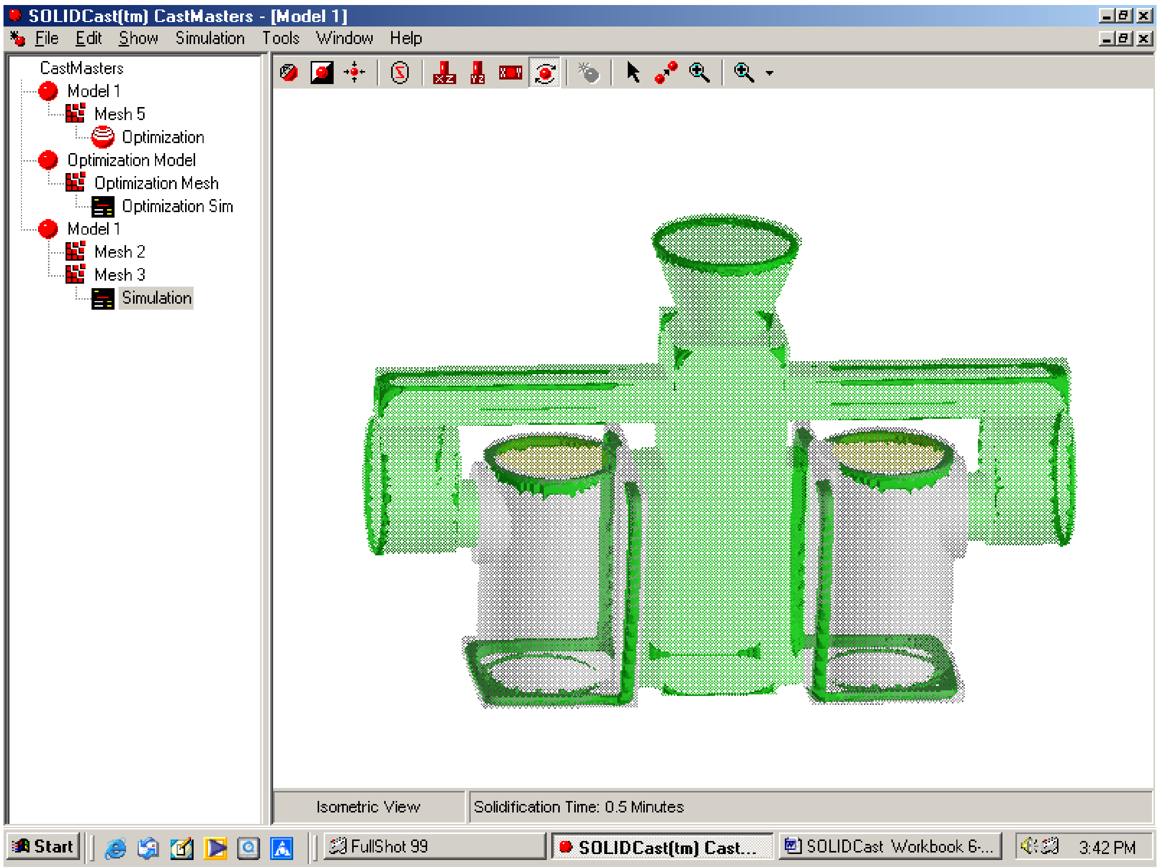
To stop the simulation, click on the button labeled Stop Sim. This will cause the simulation to stop, and the system will create data that can be plotted to show whatever results are available. Note that at this point it is possible to exit from the SOLIDCast system without losing any data. At any later point in time, it will be possible to load the project and restart the simulation to allow it to continue to completion.

To restart a simulation, double-click on the simulation icon on the project tree. You will see a window similar to the following:



By clicking on the button labeled Restart Sim, the simulation will be restarted and will run from the point at which it was interrupted.

Note that if you plot intermediate data from an interrupted simulation, the data may be incomplete and the plots may appear to be incomplete. For example, plotting Solidification Time for the above incomplete simulation shows the following result (Note: See following units for details on how to plot Solidification Time):

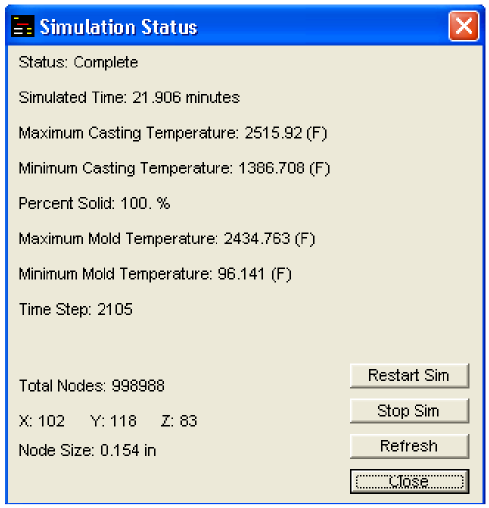


This plot obviously has some data missing, due to the fact that the simulation has not completed, and only partial data is available. However, this type of image may often be enough to establish a general idea of what is happening inside the casting, and may be used to obtain a preliminary answer as to whether a given rigging design appears to be working. Sometimes this may be enough to evaluate a design and indicate whether a redesign is necessary, before a simulation has completely finished.

UNIT 17: Plotting Results Using Iso-Surfaces

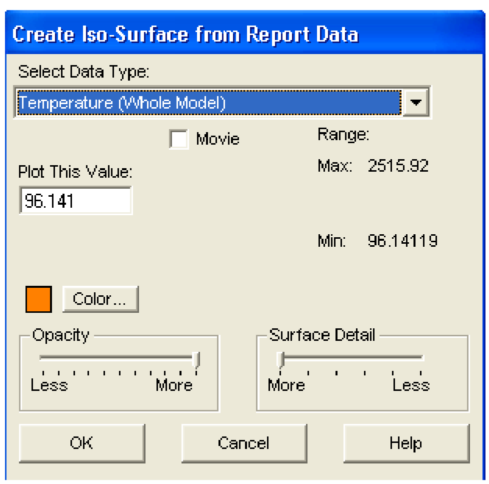
An Iso-Surface is a three-dimensional surface inside the casting model that displays where certain output data has a particular value. This means that you are plotting ONE VALUE of the selected type of output data.

To plot an Iso-Surface, first double-click on the Simulation icon on the project tree, in the SOLIDCast main screen. This will display a window similar to the following:



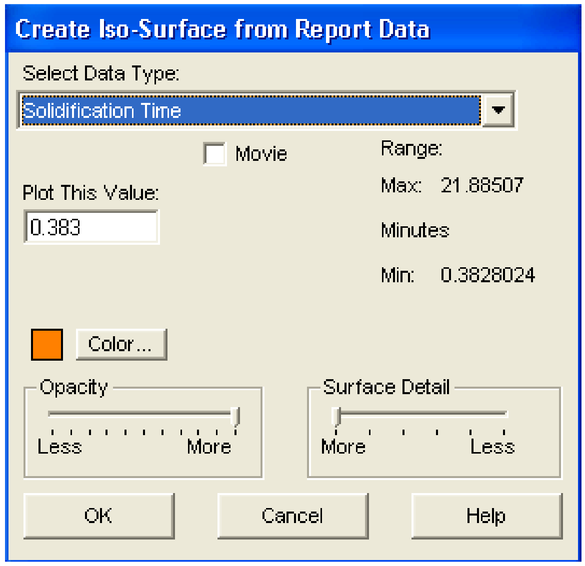
This shows some general information about the simulation that was run. Click on the Close button.

Now, at the menu bar select Simulation… Plot Iso Surface. A window similar to the following will appear:

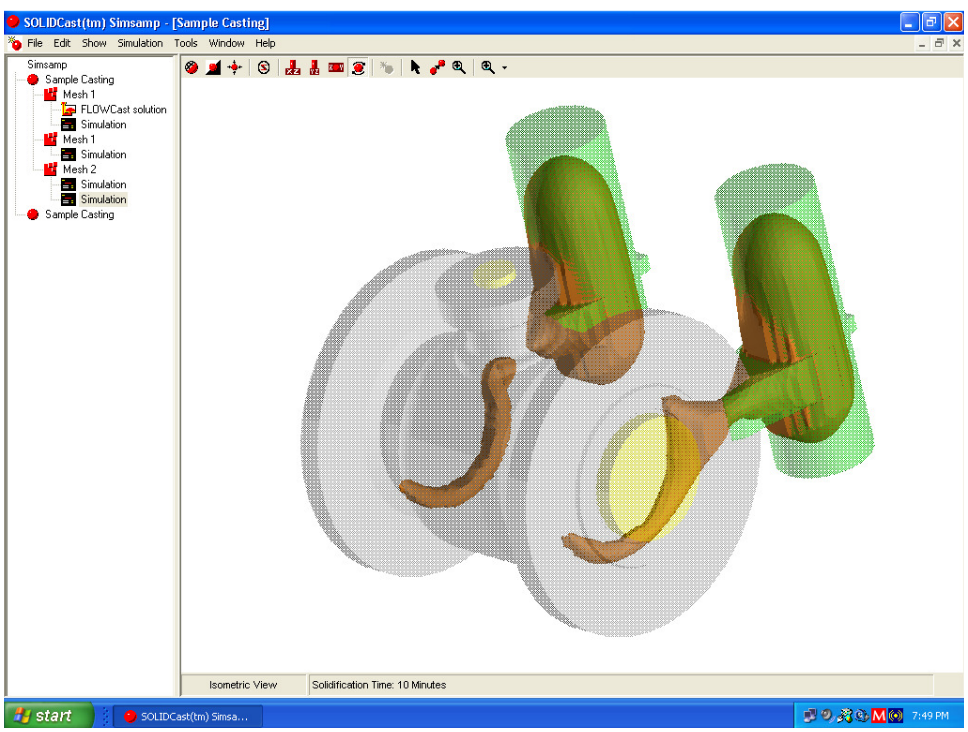


The type of data to be plotted is indicated in the window under the label Select Data Type. Click on the “down” arrow next to this field, and you will see some additional selections.

As an example, here we will select Solidification Time. When you click on this selection, the window will appear as follows:



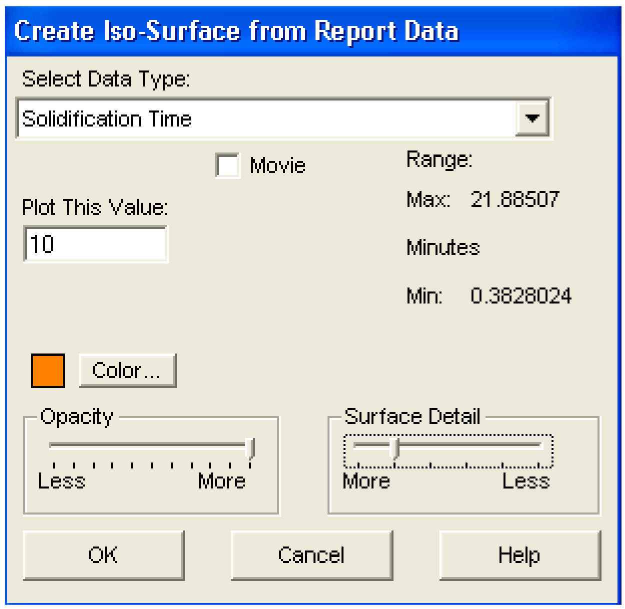
Here, the system is indicating that the minimum solidification time in the casting is .383 minutes (i.e., the first point in the casting finished freezing at .383 minutes) and the very last point solidified at 21.9 minutes. Under the label Plot This Value: is a field where we can enter a value to plot. For this example, a value of 10 minutes has been entered. At this point, you would click on OK, and the following would appear:



The shapes inside the casting represent the locations of metal that had not yet completely solidified as of 10 minutes. All of the metal outside these shapes has solidified prior to 10 minutes.

By changing and re-plotting Iso-Surfaces at various values, it is possible to get a good idea of how solidification progressed within this casting.

Note: If you have a model meshed at a high node count and there are a lot of nodes within the casting, generation of Iso-Surfaces may take a long time. It is possible to decrease this time by changing the parameter called “Surface Detail”. This creates a surface with slightly less detail, but one that takes much less time to generate. Surface Detail is adjusted as shown following:



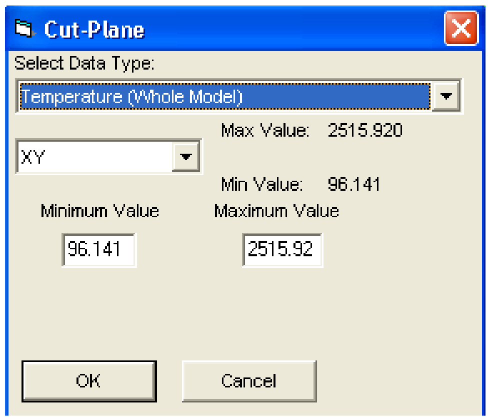
One notch of adjustment on Surface Detail will make a considerable difference in the time to create an Iso-Surface within a casting model.

Note: This plot can also be used to create an animated movie. Instructions for making movies are given in Unit 37.

UNIT 18: Plotting Results Using Cut Planes

Cut Plane plotting involves showing simulation results on a 2D plane cut through the casting.

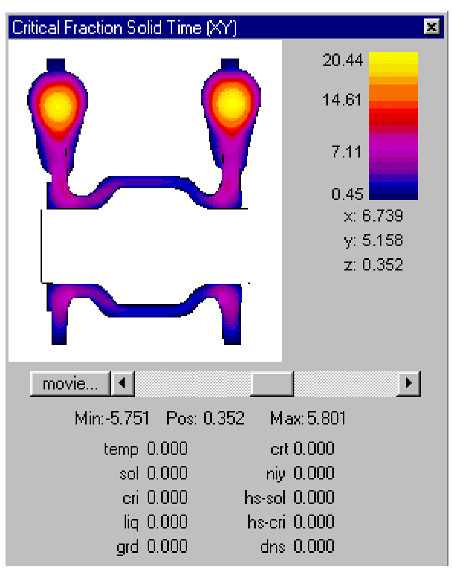
To plot a Cut Plane, first double-click on the Simulation icon on the project tree, in the SOLIDCast main screen. Now, at the menu bar select Simulation… Plot Cut Plane. A window similar to the following will appear:



You can select a cut plane in the XY, XZ or YZ orthogonal planes. Also, as with Iso-Surfaces, you can select any of the various types of output data to plot.

The system shows you the minimum and maximum values of the selected data. In the small windows you can enter plot ranges to create the cut plane plot. These plot ranges must be within the Min and Max Values as shown.

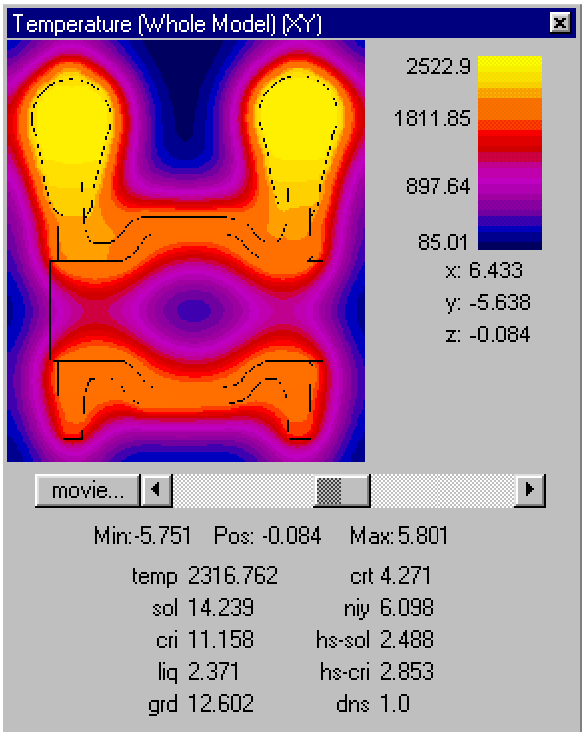
The following example shows a Cut Plane plot of Critical Fraction Solid Time within the casting and riser, in the XY orientation:



You can move the cut plane within the model, by moving the slider bar or clicking on the left or right arrows.

You can also click on a point on this image and display all of the available data about that point below the image.

Another example Cut Plane plot shows temperature distribution within both the cast material and the mold material:



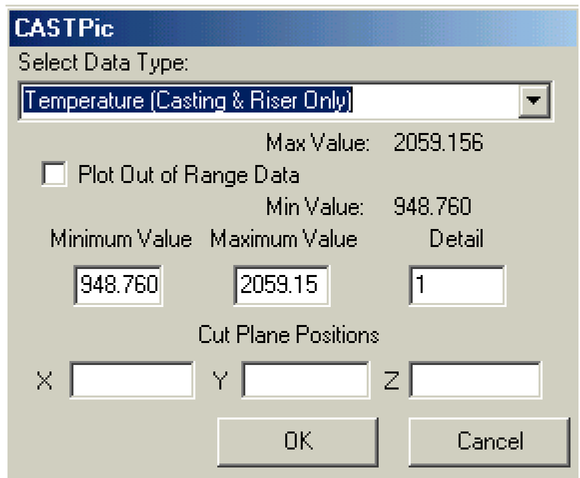
Here the diffusion of heat into the sand mold can be seen clearly.

Note: This plot can also be used to create an animated movie. Instructions for making movies are given in Unit 37.

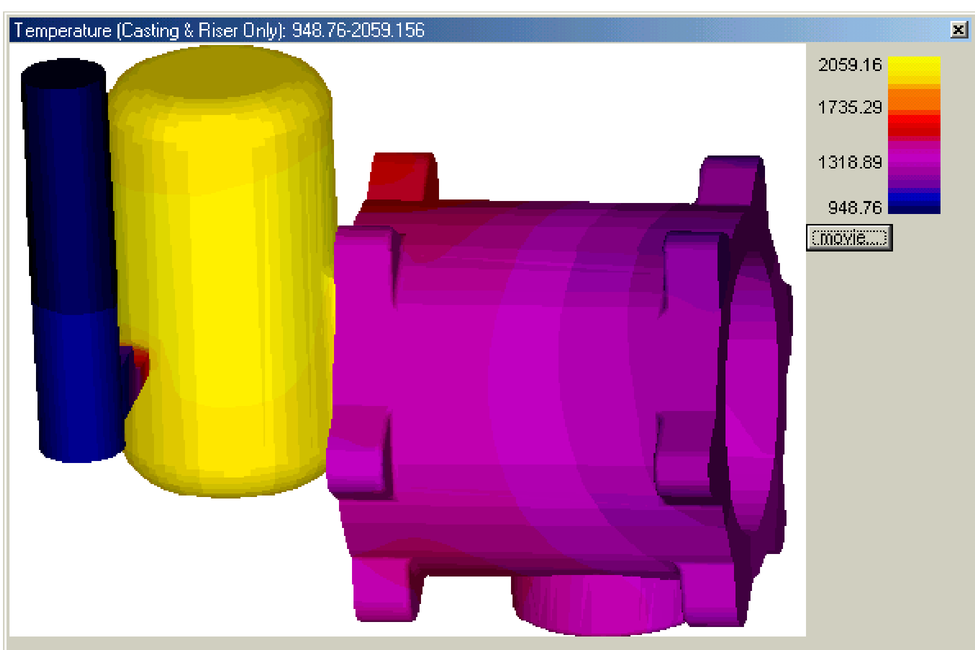
UNIT 19: Plotting Results Using CastPic

CastPic is a function that can create a detailed, three-dimensional image of the casting with result data plotted onto the image as a range of colors. This function can show the whole casting or section the image to show internal details in the casting.

To make a CastPic image, first double-click on the Simulation icon on the project tree, in the SOLIDCast main screen. Now, at the menu bar select Simulation… CastPic Plot. A window similar to the following will appear:

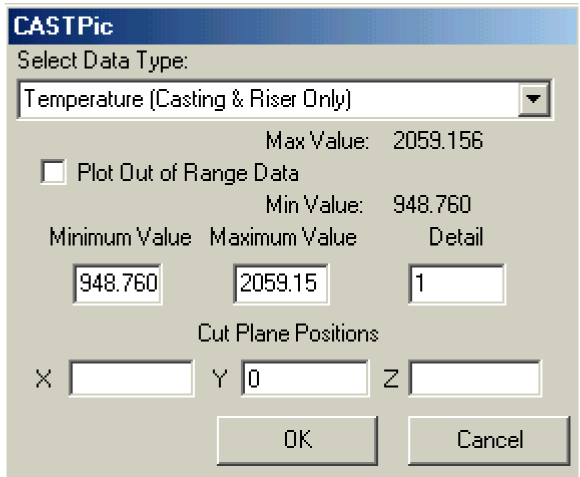


You can accept the Min and Max values as the values to plot, or enter new ones to change what the range of color in the plot indicates. Clicking on OK on the above window produces the following image showing temperature of the casting:

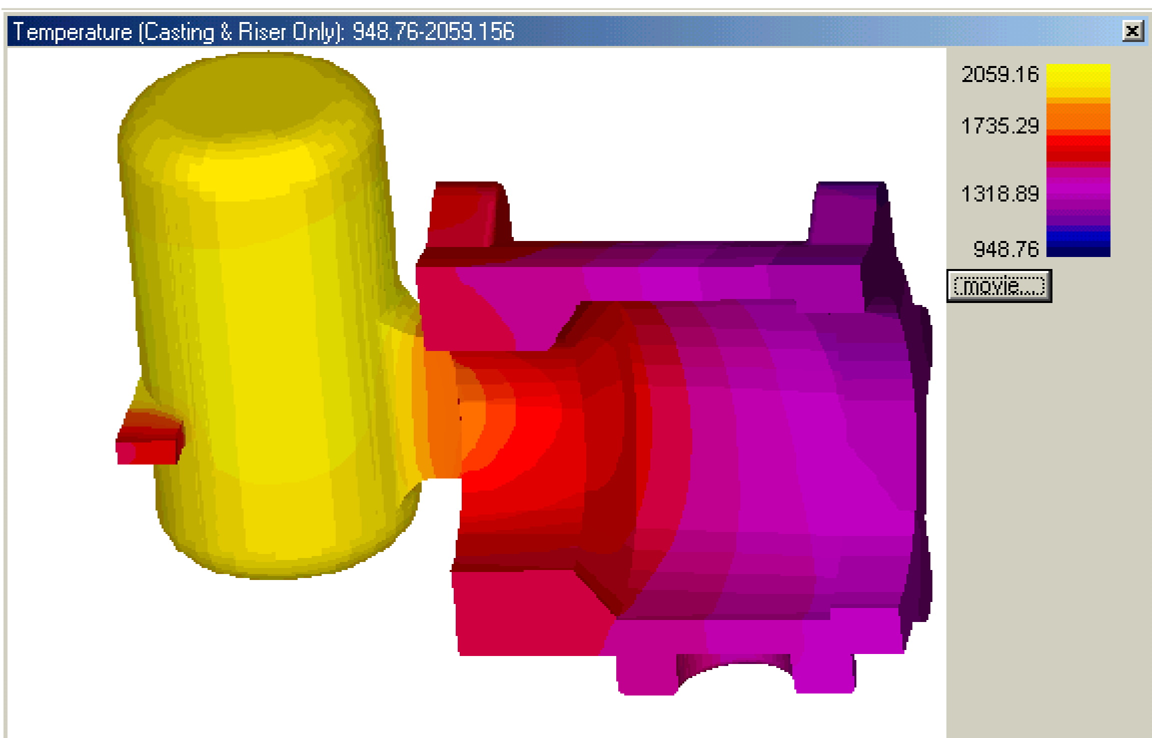


The angle of view of this image is the SAME as the angle of view of the model in the current model window.

A selection which would produce a cut plane through this model at a value of Y=0.000 would be as follows:



This would produce the following CastPic image:

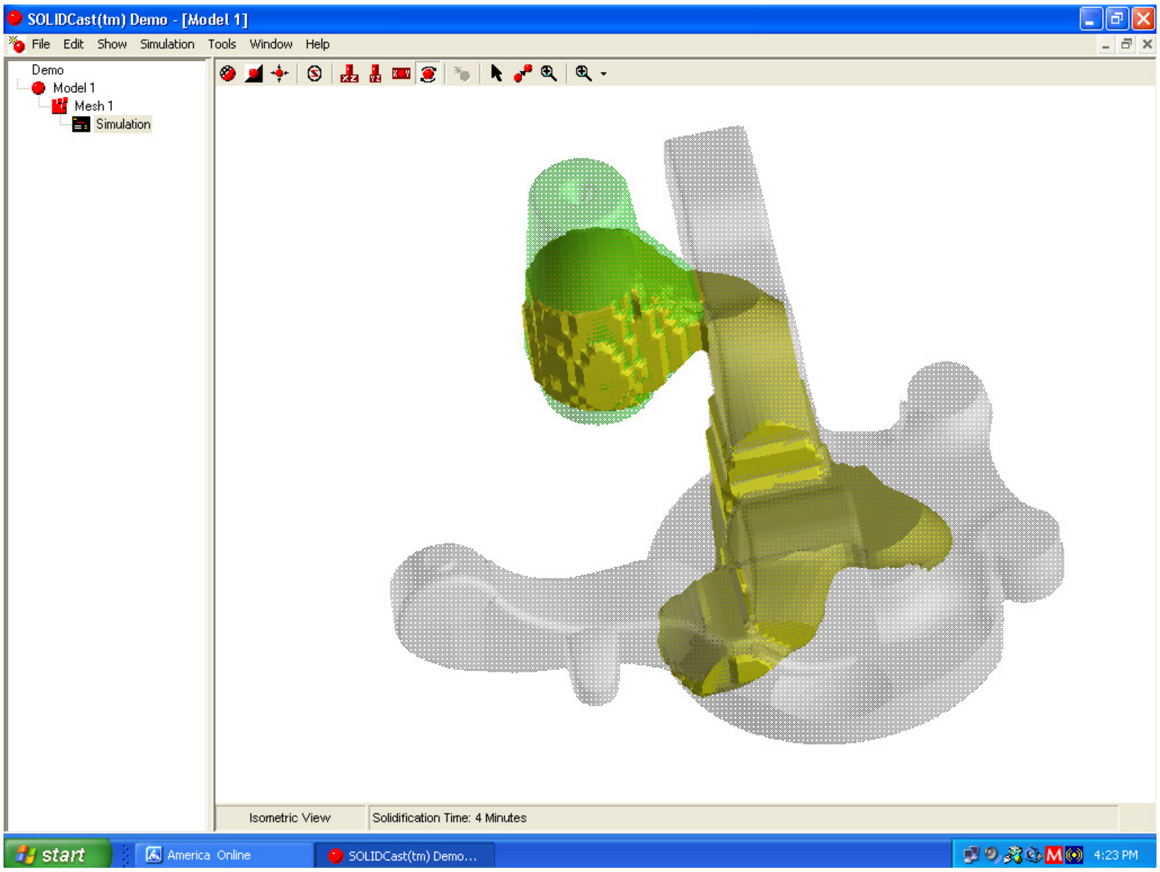


In this image, the casting has been sectioned to reveal internal details.

UNIT 20: Output Criteria: Solidification Time

Solidification Time shows the time, in minutes, for each part of the casting to become COMPLETELY solid, i.e., to cool to the Solidus Point.

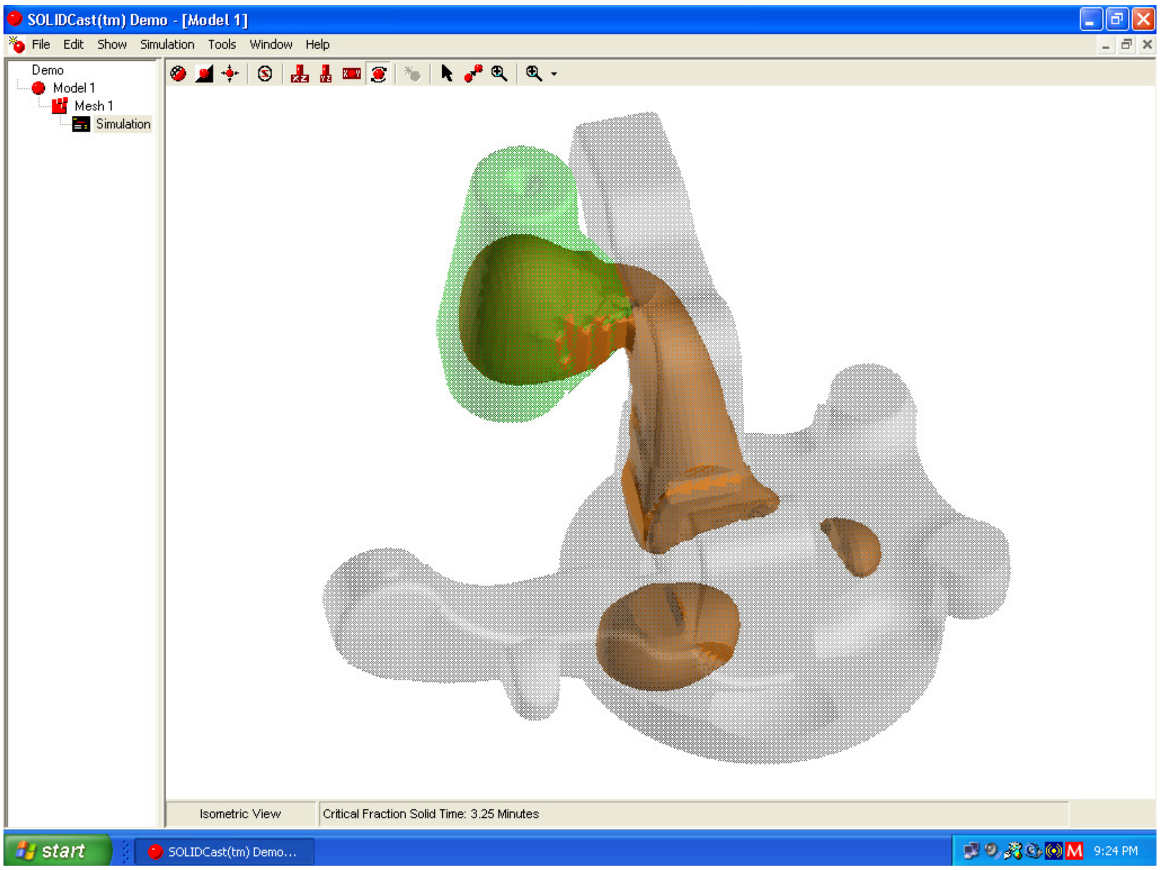
Viewing a plot of Solidification Time shows the progress of solidification through the casting. This can help to locate isolated areas of molten metal within the casting and to get a general idea of progressive solidification in various areas of the casting.



For most alloys, you are looking for a good progression of solidification, from the edges of the casting, out through the riser contacts and into the risers. If any area becomes isolated, it is an area that will be prone to shrinkage.

UNIT 21: Output Criteria: Critical Fraction Solid Time

Critical Fraction Solid Time records the time, in minutes, for each part of the casting to reach the Critical Fraction Solid Point. This is the point at which the alloy is solid enough that liquid feed metal can no longer flow. Therefore, for judging directionality of solidification, and whether any isolated areas have formed within the casting that cannot be fed by risers, Critical Fraction Solid Time is generally a better indication than Solidification Time. Plotting Critical Fraction Solid Time gives a good indication of whether any contraction that forms will be able to be fed by liquid feed metal within the risers or feeders.



The areas that appear as isolated pools of molten metal will not be able to receive feed metal from the risers if any contraction should occur during cooling and solidification.

This plot is interpreted in the same manner as Solidification Time. That is, you want a good progression from the edges of the casting, in towards the riser contacts, and out into the risers themselves.

In cast irons, which may have an expansion component to solidification, an isolated area shown in this plot MAY not exhibit shrinkage. The Material Density function should also be checked to verify the formation of shrinkage. See Unit 22 for MDF information.